

Workbench User's Guide



ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 15.0 November 2013

ANSYS, Inc. is certified to ISO 9001:2008.

Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFID-ENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Overview	1
Interacting with Project Objects	1
Workbench Tabs and Views	2
The Project Tab	4
The Toolbox in the Project Tab	4
The Project Schematic	5
Systems and Cells in the Project Schematic	6
Project Schematic Links	6
Project Schematic Workflow	8
Configuring ANSYS Workbench	9
Customizing the Toolbox on the Project Tab	9
Configuring Units in Workbench	10
Setting ANSYS Workbench Options	13
Project Management	14
Appearance	
Regional and Language Options	16
Graphics Interaction	
Journals and Logs	
Project Reporting	
Solution Process	
Repository	
Extensions	
Mechanical APDL	
CFX	
Fluent	
Mechanical	
Microsoft Office Excel Options	
Meshing	
Design Exploration Options	27
Geometry Import	
Using Software Licensing in ANSYS Workbench	
Working in the ANSYS Workbench Project Tab	
Adding Systems to the Project Schematic	
Naming and Renaming Systems	
Working through a System	
Defining your Simulation Geometry	
Basic Mechanical Analysis Workflow	
Basic Fluid Flow Analysis Workflow	
Basic Fluid Flow Analysis, Starting from Geometry	
Basic Fluid Flow Analysis, Starting from an Imported Mesh	
Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (ANSYS CFX) Sys-	
tem	43
Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (Polyflow) System	
Creating and Linking a Second System	
Creating Independent Systems	
Creating Connected Systems	
Duplicating Systems	
Moving, Deleting, and Replacing Systems	
Working in ANSYS Workbench	
Using Remote Solve Manager (RSM) in ANSYS Workbench	

iii

Submitting Projects to Remote Solve Manager (RSM) for Remote Update	
Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes	50
Submitting Mechanical Jobs to RSM	55
Submitting Mechanical Application Jobs to RSM	55
Workbench RSM Integration with the Mechanical Application	56
Submitting Mechanical APDL Jobs to RSM	
Submitting Mechanical APDL Jobs to HPC via RSM	59
Submitting Mechanical APDL Jobs to PBS via RSM	59
Submitting Mechanical APDL Jobs to LSF via RSM	60
Submitting Fluids Jobs to RSM	60
Submitting CFX Jobs to RSM	60
Submitting CFX Jobs with Design Points to RSM	62
Submitting CFX Jobs to HPC via RSM	62
Submitting CFX Jobs to PBS via RSM	62
Submitting CFX Jobs to LSF via RSM	62
Submitting Polyflow Jobs to RSM	63
Submitting Fluent Jobs to RSM	63
Solution Properties: Use Setup Launcher Settings	64
Submitting Fluent Jobs to HPC via RSM	67
Submitting Fluent Jobs to PBS via RSM	67
Submitting Fluent Jobs to LSF via RSM	68
Exiting a Project during an RSM Solution Cell Update	68
Using Journals and Scripts	
Journaling	69
Recording and Playing a Journal	70
Using the Command Window	71
Scripting	71
Project File Management	71
Project Directories	71
dp0 Subdirectory	72
user_files Subdirectory	72
dpall Subdirectory and Working with Design Points	73
Example Project	73
Working with Files and Projects	74
Importing Files	74
Archiving Projects	75
Project Recovery	75
Project Locking	76
Notes About Project File Management	76
ANSYS Workbench Files	77
Importing Legacy Databases	78
Working with the Chart View	81
Chart Types	81
Setting Chart Properties	82
Chart Zoom, Pan, and Rotate	85
Using the Triad	85
Saving a Chart	85
Working with Project Reports	86
Using Help	87
Troubleshooting	89
Problem Situations	89
Startup or Graphics Problems	90

Limitations	91
Error Messages	91
Working with Parameters and Design Points	
The Parameter Set Tab or Parameters Tab	93
Working with Parameters in the Parameter Set Tab or Parameters Tab	94
Working with Design Points in the Parameter Set or Parameters Tab	95
Parameters	96
Custom Parameters	96
Design Points	101
Updating Design Points	103
Design Point Update Order	104
Activating and Exporting Design Points	
Updating Design Points via Remote Solve Manager (RSM)	107
Aborting or Interrupting an RSM Design Point Update	111
Exiting a Project during an RSM Design Point Update	112
Note for Using Windows LSF Scheduler	112
Product-Specific Limitations	113
Reserving Licenses for a Design Point Update	114
Tracking Licenses	115
Returning Reserved Licenses	116
Using HPC Parametric Pack Licenses	116
Design Point Update Data	117
Design Point States	117
Working with ANSYS Workbench and EKM	119
Creating a Repository Connection	119
Launching EKM with a Web Browser	
Working with Existing Repository Connections	
Registering an ANSYS Workbench Session with an EKM Repository	
Working with ANSYS Workbench Projects Saved in an EKM Repository	122
Saving a Project to an EKM Repository	
Opening a Project from an EKM Repository	
Sending Project Changes to the EKM Repository	
Getting Project Changes from the EKM Repository	
Managing EKM Repository Project Changes	129
Importing Repository Files	
Troubleshooting	133
ANSYS Workbench Systems	
Analysis Systems	
Design Assessment	136
Electric	138
Explicit Dynamics	
Fluid Flow (CFX)	
Fluid Flow (Fluent)	
Fluid Flow (Polyflow)	140
Harmonic Response	
Hydrodynamic Diffraction	
Hydrodynamic Time Response	
IC Engine	
Linear Buckling and Linear Buckling (Samcef)	
Magnetostatic	
Modal and Modal (Samcef)	
Random Vibration	145

Response Spectrum	146
Rigid Dynamics	146
Static Structural and Static Structural (Samcef)	147
Steady-State Thermal and Steady-State Thermal (Samcef)	147
Thermal-Electric	
Throughflow	
Transient Structural and Transient Structural (Samcef)	148
Transient Thermal and Transient Thermal (Samcef)	
Component Systems	
Autodyn	
BladeGen	
CFX	
Engineering Data	
Explicit Dynamics (LS-DYNA Export)	
External Data	
Creating and Configuring an External Data System	
Importing a CDB File as Input	
Importing a CDB rife as input	
ANSYS External Data File Format	
Importing Multiple Data Sets	
Source Point Analytical Transformations	
,	
Performing System Coupling Simulations Using External Data	
External Data System and System Coupling Configuration	
Supported Input File Formats	
Supported Data Types	
External Model	
Creating and Configuring an External Model System	
Finite Element Modeler	
Fluent	
Fluent (with TGrid meshing)	
Geometry	
ANSYS ICEM CFD	
Elements of the ICEM CFD Component	
Creating an ICEM CFD Component	
Updating ICEM CFD Projects	
Interface Differences in the Data-Integrated ICEM CFD	
Setting Parameters	
User-Defined Parameters Example	194
Icepak	197
DesignModeler to Icepak	201
PostProcessing of Icepak Results	204
Thermal Results From Icepak to Mechanical	205
Design Explorer - Icepak Coupling in Workbench	206
Ansoft - Icepak Coupling in Workbench	
ANSYS Icepak - Workbench Integration Tutorial	
Mechanical APDL	
Mechanical Model	
Model-to-Model Connections	
Mesh	
Mesh-to-Mesh Connections	
Microsoft Office Excel	
Preparing the Excel file	
	232

Set Up the Excel Calculator	. 234
Support of Units	. 235
File management and modification of the worksheet	. 235
Limitations	. 236
Troubleshooting	. 236
Polyflow	. 236
Results	. 237
System Coupling	. 237
Solution Cell Properties	. 237
TurboGrid	. 238
Vista AFD	. 238
Vista CCD and Vista CCD with CCM	. 238
Vista CPD	. 239
Vista RTD	. 239
Vista TF	. 239
Custom Systems	. 240
Design Exploration	
External Connection Systems	
ANSYS Workbench Interface Reference	. 245
Tabs within Workbench	. 245
Views within Tabs	. 245
Project Schematic View	
Common Views	
Toolbox View	
Toolbox Customization View	
Files View	
Outline View	
Properties View	
Table View	
Chart View	
Scene View	
Solution Information View	
Persistent Views	
Messages View	
Progress View	
Cells in Workbench	
Types of Cells	
Engineering Data	
Geometry	
Model/Mesh	
Setup	
Solution	
Results	
Understanding Cell States	
Typical Cell States	
Solution-Specific States	
Failure States	
Cell Properties	
Common Cell Properties	
Menus in Workbench	
Menu Bar	
File Menu	

View Menu	262
Tools Menu	263
Units Menu	264
Extensions Menu	264
Help Menu	265
Context Menus	
Common Context Menu Options	265
Transfer Context Menu Options	
Tab Context Menu Options	269
System Header Context Menu Options	
Project Schematic Context Menu Options	
Link Context Menu Options	
ANSYS Workbench Tutorials	275
Glossary	
Index	

Overview

ANSYS Workbench combines the strength of our core simulation tools with the tools necessary to manage your projects. You will work with your ANSYS Workbench project on the main project workspace, called the **Project** tab. The project is driven by a schematic workflow, represented visually on a flowchart-like diagram called the **Project Schematic**. To build an analysis, you add building blocks called **systems** to the **Project Schematic**; each system is a block of one or more components called **cells**, which represent the sequential steps necessary for the specific type of analysis. Once you have added your systems, you can link them together to share and/or transfer data between systems.

From the cells in the **Project Schematic**, you can work with various ANSYS applications and analysis tasks. Some of these open in tabs within the Workbench environment, while others open independently in their own windows.

ANSYS applications allow you to specify parameters such as geometry parameters, material properties and boundary conditions. Parameters can be defined within the application and managed at the project-level in the Workbench environment.

To perform your analysis, you will work through the cells of each system in order—typically from top to bottom—defining inputs, specifying project parameters, running your simulation, and investigating the results.

Workbench enables you to easily investigate design alternatives. You can modify any part of an analysis or vary one or more parameters, and then automatically update the project to see the effect of the change on the simulation result.

Related Topics:

Interacting with Project Objects Workbench Tabs and Views The Project Tab

Interacting with Project Objects

In ANSYS Workbench, a **Toolbox** on the left side of the **Project** tab contains analysis systems, component systems, and other types of project objects. The **Project Schematic** view will contain the system(s) that you have added, with each system made up of one or more cells. You can interact with these project objects in a number of different ways:

- Single-click: Single-click an object to select it. This does not modify data or initiate any action.
- Double-click: Double-click an object to initiate the default action. This allows users who are familiar with ANSYS Workbench to quickly move through basic or common operations.
- Right- click: Right-click to display a context menu applicable to the current state of the selected object. From the context menu, you can select from multiple actions. The default action is shown in bold and is the action that will occur if you double-click the object.

1

Drag-and-drop: Drag-and-drop an object to preview possible locations for it on the **Project Schematic**. A
drag-and-drop operation can have multiple alternative targets, depending on context and schematic
complexity. Holding down the mouse button, hover over any target to see details of how the target location
would be implemented (for example, what components would be connected after the operation is completed).

To cancel a drag-and-drop operation, press the **Esc** key while holding down the mouse button.

Workbench Tabs and Views

When working in the Workbench environment, you will be dealing primarily with interface elements called **tabs** and **views**.

Workbench Tabs

In Workbench, *tabs* serve as workspaces that allow you to interact with different parts of your project. In addition to the **Project** tab, other tabs can be opened for other workspaces; for example, you can open the **Parameter Set** tab by double-clicking on the **Parameter Set** bar on the **Project Schematic**. Workspace tabs can be accessed either by double-clicking the associated cell, or by right-clicking it and selecting **Edit**.

Each cell that does not launch an external application has a single workspace that can be opened inside Workbench. In some cases, it may be possible to open multiple tabs of the same type; for instance, if you have three **Engineering Data** cells in three separate systems, you could have three **Engineering Data** tabs open at the same time.

Tabs in Workbench are made up of multiple **views** containing information relevant to portion of the analysis shown in the tab. Each tab also has a context-specific toolbar containing buttons for the operations that are available, given the current state of the project.

You can click on tabs to move between them or right-click them to view available context menu options such as **Close Tab**, **Close Other Tabs**, and **Close All**. You can also close individual tabs by clicking the "x" icon. Only the **Project** tab cannot be closed.

For more information, see Tabs within Workbench (p. 245).

Workbench Views

Tabs in Workbench are made up of multiple **views** that can be reconfigured according what information you want to be shown in the tab.

Some views are common to multiple tabs, while others are tab-specific. Some views are only shown in the tab for which they've been enabled, while others, once enabled, are shown across all the tabs either until you either disable them or reset the tab layout.

The configuration and persistence of the views across tabs varies to give you the maximum flexibility in customizing the information you see on each tab. The header bar of each view contains icons and context menus that allow you to control view attributes such as visibility, size, and floating/docked state. You can access the context menu either by clicking the down-arrow or by right-clicking in the header.

To open a view, select it from the View menu. For example, if you're in the Project tab and select View
 Outline, the Outline view will be shown in the Project tab. Once the view is open, it will update its

content according to the what you've selected; by selecting different objects, you can view or edit the associated properties of those objects.

- To close a view, deselect it from the View menu, select Close from the header bar context menu, or click the "x" icon in its header bar.
- To change how the view displays, use the **Minimize**, **Maximize**, and **Restore** context options.
 - When you minimize a view, it appears as a tab in the bottom left corner of the ANSYS Workbench window. You can only maximize a floating view; you cannot maximize a docked view.
 - After you have maximized a view, use the **Restore** option on the **Windowing** drop-down menu to return
 the view to its pre-maximized size and location.
- To resize a view, you can use your mouse to drag its edges to the desired size.
- You can also "float" and "redock" views. By default, most views are "docked," or embedded in their tab. By default, most views are embedded in their tab. You can choose to separate a view from its tab, so it can be moved outside of the Workbench window. When you redock a view, it returns to its place on its usual tab.

When you float a view, you separate it from its tab, so it is still available when you switch tabs. This is especially useful when you want to see a view that is not available as part of a tab. For example, the **Files** view is not visible in the **Parameter Set** tab, but you can float the **Files** view so you can see it while working with project parameters.

- To float a view, select the **Float** context option or the thumbtack icon in the view header. You can also "tear" the view from its dock by dragging the header with your mouse.
- To redock a view, select the **Redock** context option or the thumbtack icon in the view header. When you redock a view, it will return to its normal tab and may not be visible in the current tab. You can also select the **View** > **Reset Workspace** menu option, which resets the views in the current tab to their default positions.
- You can reset the views for one tab or for all the tabs.
 - \rightarrow To reset all the views in the current tab to their default position, select **View > Reset Workspace**.
 - → To reset all the views in the project,, select **View > Reset Window Layout**. This resets the views in all tabs to their default positions and opens the **Project** tab.

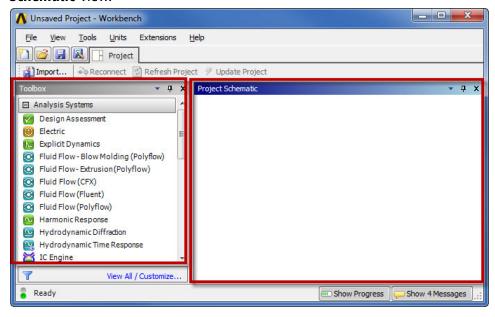
Some views—for example, the **Table**, **Chart**, **Outline**, and **Properties** views—are defined per tab, so that changes to a view are specific to that tab. For example, if you resize and float the **Chart** view in the **Parameter Set** tab and then switch to the **Response Surface** tab in a DesignXplorer system, you'll find that the **Chart** view in the new tab will note be resized and floated. (The DX **Chart** view contains different data, and so does not reflect the changes made to the **Chart** view elsewhere.) When you return to the **Parameter Set** tab, you'll see that the **Chart** view there is resized and floated, as before.

For more information on specific views, see Views within Tabs (p. 245).

The Project Tab

When you open a project in ANSYS Workbench, it opens to the **Project** tab. The **Project** tab is just one of many tabs that can be opened in Workbench, but it is the main workspace in which you will build your analysis.

Like all tabs, the **Project** tab is made up of different views that can be reconfigured according the information you want to see. By default, the **Project** tab is comprised of the **Toolbox** view and the **Project Schematic** view.



Related Topics:

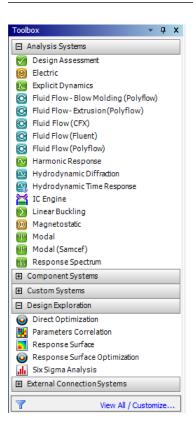
The Toolbox in the Project Tab
The Project Schematic

The Toolbox in the Project Tab

The Workbench Toolbox view contains the types of data you can add to your project.

In the **Project** tab, the **Toolbox** contains the different types of systems you can add to the **Project Schematic**. Systems are divided into categories which can be expanded or collapsed to show or hide the systems in that category. You can select systems from the following categories:

Analysis Systems (p. 135) Component Systems (p. 149) Custom Systems (p. 240) Design Exploration (p. 242) External Connection Systems (p. 243)



The contents of each category are determined by which products you have installed and what licenses are available. If you do not have a particular product installed or do not have an available license, the individual systems corresponding to that product will not be shown in the **Toolbox**. Additionally, you can customize the **Toolbox** further, specifying that only some of the available systems are visible.

The **Toolbox** view persists across tabs. It contains systems on the **Project** tab, but on other tabs it will contain other sorts of information, such as charts, engineering materials, etc.

For information on customizing the **Toolbox** on the **Project** tab, see Customizing the Toolbox on the Project Tab (p. 9).

For detailed information on system categories and individual systems, see ANSYS Workbench Systems (p. 135).

The Project Schematic

When you interact with a simulation project, you will work primarily in the **Project Schematic** view of the **Project** tab, adding systems from the **Toolbox** to the project and then working with those systems.

Projects can vary in complexity, from a single system representing all the necessary steps for a desired analysis, to a complex set of connected (linked) systems representing coupled analyses or variations in modeling approaches.

Note

It is recommended that a given project contains only systems that are relevant to a specific analysis or coupled analysis with a well-defined focus. Adding systems for multiple unrelated analyses to the same project can have an adverse effect on performance and cause corruption with portions of the project.

Related Topics:

Systems and Cells in the Project Schematic Project Schematic Links Project Schematic Workflow

Systems and Cells in the Project Schematic

Each **system** placed on the **Project Schematic** is made up on one or more analysis components called **cells**. Once all of your systems are in place, you're ready to start defining the details of your analysis. To do so, you generally interact with systems at the cell level. Right-click the system header or cell to see a menu of available options; double-click to perform the default action (bolded in the context menu).

You can interact with a cell to perform any of the following actions:

- launch an application that opens independently of Workbench
- · open a tab inside Workbench
- add connecting systems, either upstream or downstream
- · assign input or reference files
- assign properties to components of your analysis

Each cell has either an application or a tab associated with it. Some cells are associated with an application that launches in a separate window, such as Fluent or Mechanical; in some cases, multiple cells in a system can be associated with the same application. Other cells, such as the **Parameters** cell or a cell in a System Coupling system, are associated with tabs that open inside Workbench.

Note

The **Project Schematic** may reflect actions you take in applications that open independently of Workbench.

To add a system to your project, you can drag a system from the **Toolbox** and drop it on the **Project Schematic**; alternatively, you can double-click the desired system in the **Toolbox**. Once you've added systems to the **Project Schematic**, you can create links between cells to transfer and/or share data. For information on building and linking systems, see Working in the ANSYS Workbench Project Tab (p. 33).

Icons for each cell indicate the state of that particular cell—for example, whether the cell needs attention, is up-to-date, and so on. For more information, see Understanding Cell States (p. 255).

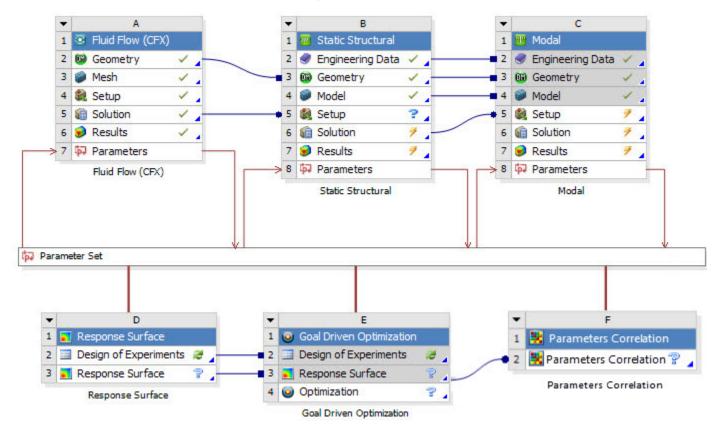
To display a quick help panel for the cell, click the blue triangle in the lower right corner of the cell (where available). The quick help message that displays will explain any immediate action that needs to be taken and may include links to more detailed help.

Project Schematic Links

Links connecting systems represent data sharing or data transfer between the systems. The primary kinds of links that may be shown in the **Project Schematic** include:

- Links indicating that data is shared between systems. These links are shown with square terminators; see the figure below.
- Links indicating data is transferred from an upstream system to a downstream system. These links are shown with round terminators; see the figure below.
- Links indicating a system is consuming input parameters. These links connect systems to the **Parameter Set** bar and are drawn with arrows going into the system, as shown in the figure below.
- Links indicating a system is providing output parameters. These links connect systems to the **Parameter**Set bar and are drawn with arrows coming out of the system, as shown in the figure below.
- Links that indicate a Design Exploration system is connected to project parameters. These links connect Design Exploration systems to the **Parameter Set** bar, as shown with the Design Exploration systems in the figure below.
- Links indicating that data is transferred from a Design Exploration Response Surface to a Parameters Correlation, as shown with systems E and F in the figure below.
- Links indicating that design point data is transferred from a Design Exploration component to a Design Exploration **Direct Optimization** system. For more information, see Transferring Design Point Data for Direct Optimization in the Design Exploration User's Guide.

Additional information on working with links can be found in Creating and Linking a Second System (p. 43) and Moving, Deleting, and Replacing Systems (p. 48).



Working with Shared Data Links

Links that are drawn with a square terminator indicate that data is shared between the two systems. Only one instance of the data exists and it is shared between the connected systems. In order to edit the details of that data, you must edit the cell on the upstream system connected via these links. In the example shown in the figure above, the **Geometry** cell from system A is shared with the **Geometry** cell in system B, which is in turn shared with the **Geometry** cell of system C. In order to edit the geometry for ANY of these systems, you must initiate the edit operation from the **Geometry** cell in system A (by double-clicking on the cell or right-clicking on the cell and selecting **Edit** from the context menu).

In many cases, it is possible to delete shared data links by right-clicking on the link and selecting **Delete** from the context menu. The data associated with the cell in the upstream system will be copied to the downstream system so the cells can be edited independently.

In some cases, you will not be able to delete links. In these cases, the linked cells in the downstream system will be shown with a gray background (as shown in systems C and E, above).

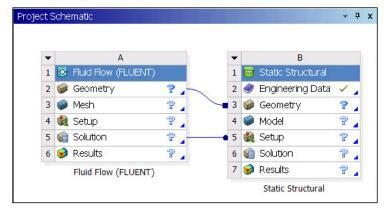
Project Schematic Workflow

To complete your analysis, you will work downward through each cell in order. In general, data flows **downstream** (from top-to-bottom within systems and from left-to-right across systems). Output data from upstream component cells is provided as the input data for downstream cells. Output data from certain types of cells in one system can also be transferred and/or shared with cells in another system.

The following example shows two systems in the **Project Schematic**, a **Fluid Flow (Fluent)** system (system A) and a **Static Structural** system (system B). In this example:

- The geometry from the **Geometry** cell in system A becomes the input for the **Mesh** cell that is downstream in that system. The mesh generated in the **Mesh** cell of the system, in turn, becomes input to the downstream **Setup** cell, and so on.
- There is also data-flow between ystem A and system B, as follows:
 - They share the same geometry, as indicated by the connector with the square terminator between the **Geometry** cell in system A and the **Geometry** in system B.
 - Solution data from System A is provided to the **Setup** cell of system B, as indicated by the connector with the round terminator.

At a glance, you can see the data relationship between the two types of analysis systems.



Configuring ANSYS Workbench

Information about configuring ANSYS Workbench can be found in the following sections:

Customizing the Toolbox on the Project Tab Configuring Units in Workbench Setting ANSYS Workbench Options Using Software Licensing in ANSYS Workbench

Customizing the Toolbox on the Project Tab

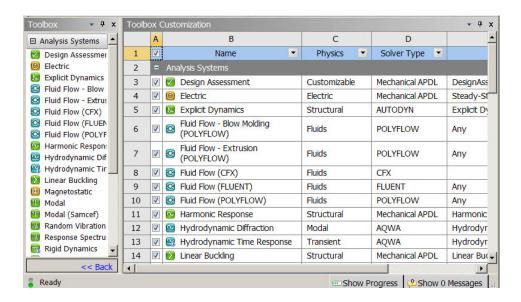
When you open ANSYS Workbench, the **Toolbox** view on the **Project** tab contains different systems you can add to your project, divided into five categories:

Analysis Systems (p. 135) Component Systems (p. 149) Custom Systems (p. 240) Design Exploration (p. 242) External Connection Systems (p. 243)

For descriptions of system categories and individual systems, see ANSYS Workbench Systems (p. 135).

In the **Toolbox** view, the specific systems included in each category depend on what products you have installed and the availability of licenses. If you do not have a product installed or do not have an available license, the systems corresponding to that product cannot be displayed in the **Toolbox**. To get more information on a product that is discussed in the documentation but you do not see in your **Toolbox**, go to the Support page of the ANSYS Customer Portal and submit an online support request. For further information about tutorials and documentation on the ANSYS Customer Portal, go to http://support.ansys.com/docinfo.

The **Toolbox Customization** view, accessed via the **View All / Customize** button at the bottom of the **Toolbox**, shows a list of all the systems available in your installation of ANSYS Workbench. You can specify whether available systems will be visible in the **Toolbox**, customizing the display to show only the systems you use frequently. Select systems to show them in the **Toolbox**, or deselect systems to hide them. When you're finished, click **<< Back** to close the **Toolbox Customization** view.



Configuring Units in Workbench

ANSYS Workbench provides the following functionality for unit systems:

- · a set of predefined unit systems that define most commonly used quantity units
- · the ability to define custom unit systems based on the predefined unit systems
- the ability to display the following project data in project unit system:
 - engineering data
 - parameters
 - charts
- the ability to share the unit system between different users via Import and Export options

Note

Unit settings in ANSYS Workbench are not passed to Fluid Flow analysis systems; to CFX, Fluent, **Results**, or TurboGrid systems; or to FSI: Fluid Flow custom systems.

To access the **Unit Systems** dialog box, choose **Units>Unit Systems** from the menu bar. You will see the following:



The following options are available from the **Unit Systems** dialog box:

Option	Description
Active Project	Sets unit system for active project.
Default 📮	Sets default unit system. This will be default unit system for every project.
Suppress/Unsuppress	Hides/displays unit system menu item. Only 15 unit systems can be unsuppressed and displayed as menu selections.
Duplicate	Creates a custom unit system based on selected unit system.
Delete	 Deletes unit system. The following unit systems cannot be deleted: Predefined Unit System Active Project Unit System Default Unit System
Import	Imports units xml file (* . xml).
Export	Exports unit system in units xml file (* . xml) format.

Units

A unit system is a collection of the preferred unit for the base, common, and other quantity types.

- · Base Units: All other units are derived from these units.
 - Angle
 - Chemical Amount
 - Current
 - Length

- Luminance

- Mass Solid Angle - Time Temperature · Common Units: These are units which are derived from the base units and are typically used as base units for other units. - Electric Charge Energy - Force - Power - Pressure Voltage Other Units: Several other units are derived from base and common units. For detail description on how units are used in expressions, see Expressions, Quantities, and Units (p. 98). **Predefined Unit Systems** ANSYS Workbench offers the following predefined unit systems: • Metric (kg, m, s, °C, A, N, V) (default unit system) • Metric (tonne, mm, s, °C, mA, N, mV) • U.S. Customary (lbm, in, s, °F, A, lbf, V) • SI (kg, m, s, K, A, N, V) • U.S. Engineering (lbm, in, s, R, A, lbf, V) You cannot edit or delete predefined unit systems. ANSYS Workbench also provides the following additional unit systems. These are suppressed by default. • Metric (g, cm, s, °C, A, dyne, V) Metric (kg, mm, s, °C, mA, N, mV) Metric (kg, μm, s, °C, mA, μN,V) Metric (decatonne, mm, s, °C, mA, N, mV)

U.S. Customary (lbm, ft, s, F, A, lbf, V)

- Consistent CGS
- Consistent NMM
- Consistent µMKS
- Consistent BIN
- Consistent BFT

For a detailed description of unit systems for the Mechanical application, see Solving Units.

You can also display values as defined and display values in project units. See Units Menu (p. 264) for more information on using these options.

Custom Unit Systems

Use the Duplicate option to create a custom unit system based on a predefined unit system.

- The default name for the new **Unit** system is **Custom Unit System**. You can change the name.
- You can change the units for any quantity type from the available list.
- The list of units that are available are either consistent with SI or US Customary, depending on the original unit system. This to assure that a consistent unit system can be constructed for solution purposes.
- Changing base units can automatically change **Common** and **Other** derived units if appropriate. For example, if the mass unit is kg, the length unit is m, and the force unit is N, then changing mass to g and length to cm will automatically change force to dyne.

Setting ANSYS Workbench Options

Select **Tools** > **Options** to set your preferences for ANSYS Workbench. The preferences you set here are local settings, affecting only you. Some changes made via the **Options** dialog will take place immediately, while others won't take place until you start a new session.

Use the **Restore Defaults** button to reset the settings visible on the current page to their default values; settings on other pages will remain unchanged.

You can set user preferences in the following categories:

Project Management

Appearance

Regional and Language Options

Graphics Interaction

Journals and Logs

Project Reporting

Solution Process

Repository

Extensions

Mechanical APDL

CFX

Fluent

Mechanical

Microsoft Office Excel Options

Meshing
Design Exploration Options
Geometry Import

Project Management

Specify the following project management options:

File Locations Specify default folders for permanent and temporary file locations. You can specify two settings:

Default Folder for Permanent Files

Controls the location where a project save/open will occur. When you choose a different location for a project via a dialog in the user interface, ANSYS Workbench remembers the folder location for subsequent operations for the rest of the session; however, upon starting a new session, the default will be the location specified here.

Folder For Temporary Files

Controls where most temporary files are written. The directory specified here holds project files that are generated before the project is saved. Once a project is saved, files are written to the project directory (seeProject Directories (p. 71) for more information).

Startup You can specify the following startup options:

- Load News Messages: Indicates if the News messages should be loaded when ANSYS Workbench is started. News messages will appear in the Messages view. The default is to load messages at startup. You must also specify the maximum age, in days, of news messages to load. You can display the messages at any time by using the Show Messages button in the status bar or by choosing View> Messages.
- **Custom RSS Feed Address**: Allows you to specify the URLs of feeds that are shown in addition to the default ANSYS news feed. You can specify multiple URLs by separating each with a semicolon (;).

Note

The **Load News Messages** and **Custom RSS Feed Address** options will be disabled if RSS feeds were disabled during the product installation.

- Start Remote Solve Manager: Indicates that the Remote Solve Manager (RSM) should be started when ANSYS Workbench is started. Defaults to disabled (so that Remote Solve Manager is not launched at startup). If enabled, RSM will be started when ANSYS Workbench is started, but the RSM interface will not be visible. On Windows, an RSM icon () will appear in the Windows System Tray. For detailed information on running RSM, please see the Remote Solve Manager User's Guide help.
- **Show Getting Started Dialog**: Displays the Getting Started dialog box on startup. The default is to display the **Getting Started** dialog at startup.

Project Archive Project Archive. Allows you to specify your preferences for archiving projects.

• .wbpz Compression Level: Allows you to specify a file compression level for archiving projects to the .wbpz format. Defaults to 3. Possible values are 0 through 9, with 0 as no compression and 9 as maximum compression. For more information on archiving, see Archiving Projects (p. 75).

Appearance

Define the appearance of your workspace, including background graphics, colors, and display defaults.

Graphics Style Controls the graphics style of the ANSYS Workbench, DesignModeler, Meshing, and Mechanical applications.

- **Background Style**: Sets a solid graphic background or a gradient background that varies from top to bottom, left to right, or diagonally. The default is the top to bottom gradient.
- **Background Color**: Sets a graphic background color from the built-in color palette. The default color is blue.
- **Background Color2**: Sets a second graphic background color from the built-in color palette. The second color is used for gradient background displays. For example, if you want a top-bottom gradient that starts out white and ends up black, **Background Color** should be set to white and **Background Color2** should be set to black. The default color is white.
- Text Color: Sets the color of all text from the built-in color palette. The default color is black.
- Edge Thickness: Sets the relative thickness display of all edges to Thin (default), Medium Thick, or Thick provided the View menu is set to either Wireframe or Shaded Exterior and Edges, and Edge Color Option is not set to Body Color.
- **Edge Colors**: Sets the colors for the **Graphics Options** feature from the built-in color palette. Default colors are presented in the table below:

Edge Type	Default Edge Color
Free	Blue
Single	Red
Double	Black
Triple	Pink
Multiple	Yellow

Meshed: Sets the color of all meshed edges from the built-in color palette. The default color is black.

Display Controls how ANSYS Workbench displays information.

- **Number of Significant Digits**: Sets the number of digits that appear for numbers throughout Workbench. The default is **5** and the range is from **3** to **10**. This setting affects only the numbers that are displayed. It does not imply any numerical round-off of internal calculations.
- Number of Files in Recently Used Files List: Sets the number of files that will appear in both the File
 menu and the context menus' Recently Used Files lists. The default is 4 files and the maximum number
 of files that you can display is 20. If the number specified here exceeds the number of recently used files
 that are available, the list will show the available number. This setting is applied to the current ANSYS
 Workbench session.
- **Beta Options**: Allows testing of unreleased ANSYS Workbench features. If selected, beta features will be displayed with the word beta in parenthesis. The default is to not show beta features. Beta features remain untested in this release and therefore are neither documented nor supported and may result in unpredictable behavior.

- **Text on Toolbars**: Allows you to turn the text labels on the toolbars on or off. Labels are on by default. This option applies to the ANSYS Workbench interface, the Mechanical application, the Meshing application, FE Modeler, and DesignModeler only.
- Connections are Bundled at Startup: Shows connections between systems as a single link. The label will indicate all bundled connections. For example, linked systems that shared Engineering Data, Geometry, and Model cells would show a single link with the notation "2:4", indicating that cells 2 (Engineering Data) through 4 (Model) are connected. Default is off. This setting defines the initial default for all projects and can be overridden within each project using View> Show Connections Bundled, or by selecting this option from the context menu on the schematic. Use of this option affects only newly created projects.
- System Coordinates are Shown at Startup: Shows the system label letters and numbers. Default is on.
 This setting defines the initial default for all projects and can be overridden within each project using
 View> Show System Coordinates or by selecting this option from the context menu on the schematic.
- Quick Help Icons in System Cells: Shows the quick help icon in cells where quick help is available. The icon appears as a small blue triangle in the lower right corner of the cell. Default is on.
- Word-wrap Text in Tables: Allows text in tables to wrap within the cells. Default is on.

Regional and Language Options

Specify the following regional and language options:

Language Allows you to specify the language to be used for ANSYS Workbench text and messages.

You can choose to use English, German, French, or Japanese, if localized files are available. ANSYS Workbench defines the language via the languagesettings.txt file, the language selected at installation, or the language specified via this option. You can also manually edit the languagesettings.txt file to specify the language. If you manually edit the languagesettings.txt file, the ANSYS Workbench user interface will display in the language you specify upon startup.

You must exit ANSYS Workbench and start a new session for changes to the language option to take effect.

ANSYS Workbench will look for the languagesettings.txt file in the following locations, in order:

- 1. %appdata%\Ansys\v150 on Windows or \$HOME/.ansys/v150 on Linux
- 2. <install_dir>\Ansys Inc\v150\commonfiles\Language (Windows) or <install_dir>/ansys_inc/v150/commonfiles/language (Linux)

Be aware that not all external applications may be translated; therefore, you may see some applications' text and messages in English even if you have specified a different language.

Graphics Interaction

Specify project management options, including mouse button behavior, your pan-zoom-rotate behavior, and selection defaults.

The **Mouse Button Assignments** category includes options for setting the various button controls on the mouse as well as button combinations with the **Shift** and **Ctrl** keys.

The **Pan, Rotate and Zoom** category's **To Zoom in Closer** option allows you set the preference of whether to zoom on a model by moving the mouse in or out. With **Dynamic Viewing**, if you make a standard view change (such as front, back, left, right, bottom, top, isometric, and Look At Face/Plane/Sketch), a short animation shows the model moving/twisting toward its final pose. Turn off dynamic viewing if you are using an older graphics card. The **Use Spaceball** setting enables the use of the Spaceball 3D import device (not supported in UNIX).

The **Selection** category's **Extend Selection Angle Limit** sets a limit in degrees for what kind of face and edge angles the system considers "smooth". This affects the Extend to Adjacent and Extend to Limits Extend Selection toolbar buttons in DesignModeler. Extend Selection buttons are also present in the Mechanical application. The default value is 20° and the range is from 0° to 90°. The **Angle increment for configure tool** sets the angular increment of the Configure tool when defining a joint.

Journals and Logs

Specify whether to create journal and log files and options associated with each.

Journal Files Allows you to specify your preferences for journal files.

- **Record Journal File**: Specify if a journal file should be recorded. If this check box is selected, ANSYS Workbench writes a journal file (.wbjn) for each ANSYS Workbench session and the following options are enabled:
 - **Journal File Directory**: Select the directory to which journal files should be written.

On Windows, the directory preference defaults to %TEMP%\WorkbenchJournals.

On Linux, the directory preference defaults to the following, in order:

- 1. \$TEMP/WorkbenchJournals
- 2. \$TMP/WorkbenchJournals
- 3. \$HOME/.ansys/WorkbenchJournals

If you record a session (**File> Scripting> Record Journal**) for future playback, the same information is written to both the location specified here and to the file specified when you begin recording.

- Days to Keep Journal File: Specify the number of days to keep journal files. Defaults to 7 days.
- When running a journal file, pause after each command: Specify whether there should be a pause after each command when running a journal file. If this check box is selected, the following option is enabled:
 - Seconds to Pause: Specify the number of seconds to pause between commands. Defaults to 1 second.
- Include journal comments for newly created objects: Select to specify that journal comments should be included for newly created objects. For each command or transaction in the journal, the journal will include a set of comments with a query providing the data reference for each object created as a result of that command or transaction. If more than 10 objects of the same type are created, individual queries will not be created; the journal will include a statement indicating the number, type, and name indices of the objects created.

If this check box is selected, the following option is enabled:

 Include journal comments for properties of newly created: Select to specify that for each object query, the journal will include a set of comments showing the values of all visible properties for the object.

Journal comments for newly created objects will also be displayed in the Command Window, when it is active.

Workbench Log Files Use these settings to specify your preferences for Workbench log files.

• Write Workbench Log Files: Specify whether Workbench log files should be written. If this check box is selected, ANSYS Workbench writes two log files for each ANSYS Workbench session. The two files are:

```
UIEventsprocID.log
CoreEventsprocID.log
```

where procID is the process ID of the ANSYS Workbench session. To identify the log file of the most recent session, it is most convenient to sort the directory by date. The log file will contain information that is useful to technical support whenever an error is encountered.

Other applications may continue to write log files to their own directories, even if launched from ANSYS Workbench.

When this check box is selected, the following options are enabled:

 Workbench Log Files Directory: Select the directory to which Workbench log files should be written.

On Windows, the directory preference defaults to %TEMP%\WorkbenchLogs.

On Linux, the directory preference defaults to the following, in order:

- 1. \$TEMP/WorkbenchLogs
- 2. \$TMP/WorkbenchLogs
- 3. \$HOME/.ansys/WorkbenchLogs

The log file directory preference takes effect immediately when the **Options** dialog is closed.

Days to Keep Workbench Log File: Specify the number of days to keep Workbench log files. Defaults to 7 days.

Project Reporting

Specify the following project reporting options:

Use these settings to control project reporting behavior. Select **After exporting report, automatically open in default browser** to launch your default browser and load the report immediately upon generation. If you do not select this option, you will need to navigate to the report file (in user_files in the project directory by default) and open the .html/.htm file manually after exporting the report.

Solution Process

Use these settings to control the defaults for the **Solution Process** properties where available on **Solution** cells. For projects with parametric design points, these settings also control the defaults for the **Design Point Update Process** setting for the **Parameter Set** bar. For more information on using the **Solution Process** properties to submit a job to RSM, see Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes (p. 50).

Specify the following solution process options:

Default Update Option You can choose:

- **Run in Foreground** -- This is the default manner of running, where solutions are run within the current ANSYS Workbench session.
- **Run in Background** -- Allows you to run the solution in the background on the local machine. This option is only available for **Solution** cells which support background execution.
- **Submit to Remote Solve Manager** -- Allows you to run the solution in the background either on the local computer or on one or more different computers. If you submit to RSM, the solution will continue to run even if you close the current ANSYS Workbench session.

If you select **Submit to Remote Solve Manager**, you also have the following options:

- Default Solve Manager -- The solve manager is the machine that is configured to manage the
 queues and compute servers to be used for the remote solution. Any Solve Managers that are
 defined for RSM will appear in the drop-down list. You can choose one of those Solve Managers
 or you can specify the name of another machine.
- **Default Queue** -- You can select from the queues that you have already defined for RSM.
- Download Progress Information -- Controls whether the solver monitor periodically queries RSM for output files in order to display progress (where applicable).
- Default Progress Download Interval -- Specifies the periodic time interval with which the solver should query RSM for output files in order to display progress. Default is 30 seconds. Setting this value to zero (0) results in continuous queries; that is, as soon as files are downloaded from the compute server, ANSYS Workbench will immediately query again. This option is available for CFX systems only.
- Default Execution Mode -- Specifies if you want the solution to run in serial or parallel mode.
 Default is serial. The parallel option is available only if the selected solver supports parallel execution mode.
- Default Number of Processes -- If parallel execution mode is selected, allows you to specify the number of processes to use if you choose to run in parallel mode. Must be set to 2 or greater.
- Pre-RSM Foreground Update -- Indicates if you want to do a local geometry-only update prior to submitting design point update to RSM. Select **Geometry** to update your geometry locally before submitting design point updates to RSM.
- Default Job Submission -- Determines how design point updates are handled if submitted to RSM. Select one of the following options:
 - → One Job for All Design Points -- All design points are submitted as a single job to RSM.

- → One Job for Each Design Point -- Each design point is submitted as a separate job to RSM (simultaneous parallel updates).
- → Specify Maximum Number of Jobs -- Design points are divided into groups and submitted in multiple jobs, up to the specified maximum number of jobs. (You can look at the RSM List view to determine which design points are assigned to each job.) If you select this option, the Maximum Number of Jobs property is enabled, allowing you to specify the maximum number of jobs that can be created.

If you have not configured Solve Managers or Queues, please refer to the RSM Overview in the Remote Solve Manager User's Guide for more information.

Default Design Point Update Order Specify the order in which design points are updated. By default, design points are solved in the order in which they appear in the **Table of Design Points**. As each design point is updated, it begins from the parameter values associated with DPO. When updating design points, it may be more efficient to update design points starting from parameter values from the previous design point, rather than starting from DPO (Current) each time. This option allows you to specify the starting condition for each design point.

- **Update from Current (DP0)** Causes each design point to be updated starting from DP0 (the default).
- **Update design points in order** Causes each design point to be updated starting from the previous design point.

If you selected **One Job for Each Design Point** in the **Default Job Submission** field, the **Default Design Point Update Order** is applicable ONLY if you also selected **Geometry** in the **Pre-RSM Foreground Update** field.

This setting will take effect the next time you create a new project. See Updating Design Points for more information.

Show Advanced Solver Options Makes the **Interconnect** and **MPI Type** Parallel Run Settings available for Fluent RSM runs. Note that these settings require you to ensure that the remote Compute Servers can accept the Interconnect and MPI Type that you specify; there is no automatic checking for such compatibility.

Exported Design Point Allows you to specify how an update will be performed for exported design points. Select one of the following options:

- **Update parameters** -- Only parameters are updated for exported design points. Default value.
- **Update full project** -- Full project is updated for exported design points.

Repository

Specify the following settings for managing simulation data in an ANSYS Engineering Knowledge Manager (EKM) repository.

Check for update upon opening a repository project

- Specify whether Workbench, upon opening a project that is saved to an EKM repository, will check for changes to the project.
- Possible values are Always Ask, Always Check, and Never Check.

If you select the Save my choice and don't ask this question again check box on the Check Repository for the Project dialog, the user preferences specified on the dialog are updated to this
drop-down.

Check for update of imported repository files upon opening a project

- Specify whether Workbench, upon opening a project containing files that are saved to an EKM repository, will check for changes to the files.
- Possible values are Always Ask, Always Check, and Never Check.
- If you select the Save my choice and don't ask this question again check box on the Check Repository for Imported Files dialog, the user preferences specified on the dialog are updated to this drop-down.

Send project changes to repository upon closing a project

- Specify whether Workbench, upon closing a project that is saved to an EKM repository, will send project changes to the repository.
- Possible values are Always Ask, Always Send, and Never Send.
- If you select the Save my choice and don't ask this question again check box on the Send Project
 Changes to Repository dialog, the user preferences specified on the dialog are updated to this dropdown.

Extensions

The **Extensions** options allow you to specify extension-handling settings for the ANSYS Workbench Customization Toolkit (ACT). Specify the following extensions options:

Under **General Options**, the following options are available:

Additional Extension Folders By default, ACT searches the user's **Application Data** folder. Define additional folders in which ACT will search from extensions in order to expose them to the **Extension Manager**. The **Extension Manager** contains any extensions located in these folders.

Save Binary Extensions with Project Specify if extensions should be save when the project is saved. Select from the following options:

- **Never**: The current loaded extensions are not saved within the project.
- **Copied but locked to the project**: The extensions are saved within the project, but are limited to that project.
- Always: The extensions are not saved within the project with no restrictions as to their use in other projects.

Under **Development**, select the **Debug Mode** check box to activate debugging mode in the Mechanical application.

For more detailed information on extensions options, see the ANSYS Workbench Customization Toolkit Developer's Guide in the ANSYS Customization Suite.

Mechanical APDL

Specify the following options for the Mechanical APDL application:

- Startup command line options
- · Default memory sizes
- · Number of processors
- · Default jobname
- Default license level
- Graphics device
- Start.ans usage
- Custom executable path
- Download Distributed Files

These items are described in detail in the ANSYS Launcher > File Management Tab documentation in the *Operations Guide* for the Mechanical APDL application.

In addition to the above Mechanical APDL application options, you can also specify the **GPU Accelerator** option. The **GPU Accelerator** option provides access to the Graphics Processing Unit (GPU) acceleration capability offered in the **Project Schematic** and inside Mechanical APDL. Three options are available, None, NVIDIA and Intel. By default None is selected. If NVIDIA or Intel is selected, specify the **Number of GPU's per Machine** in the row below. By default this number is set to 1. Choosing this option from **Options**->Mechanical APDL will apply the setting to all newly added systems in the **Project Schematic**. You can override these settings by changing the **GPU Accelerator** selection on individual systems by doing a right mouse click on **Analysis** and editing the properties.

CFX

Specify the following options for the CFX application:

Set the default initialization option for the solution cell

- The option **Update from Current Solution Data if possible** (the default) causes CFX-Solver to use the previous solution (if this exists) as its initial conditions.
 - If there is no previous solution run, CFX-Solver will use either an upstream **Solution** cell, the Initial Values definitions on the **Define Run** dialog of CFX-Solver Manager, or the initial conditions from the current state of the **Setup** cell.
- The option Update from Initial Conditions causes CFX-Solver Manager to use either an upstream Solution cell, the Initial Values definitions on the Define Run dialog of CFX-Solver Manager, or the initial conditions from the current state of the Setup cell.

Set the default update option for the solution cell

Controls whether the update proceeds as a **Background Task** or a **Foreground Task** (the default).

- During a foreground update, the user interface strictly limits what you can do (you cannot edit other cells, save, quit, etc.)
- During a background update, the user interface allows other operations and updates on other cells to take place, and you can also save and quit the project.

After saving and quitting, the solver run will still continue. You can re-open the project and use the **Reconnect** button to access data that was put into batch mode.

Upon opening a project, ANSYS Workbench automatically retrieves the data for any completed background update without the need for you to press the **Reconnect** button. If you decline to save the project before exiting, this solver data will be discarded and will not be accessible in future ANSYS Workbench sessions.

A foreground update that is in progress can be made into a background update by using the **Switch Active Solution to Background** context menu option. This action will not change the **Update Option** setting for the next run.

Set the default execution control conflict option for the Solution cell

If you add or change Execution Control in ANSYS CFX-Pre in a way that conflicts with the execution control settings stored in the **Solution** cell, by default an error message appears when you attempt to update the **Solution** cell. You can modify this behavior as follows:

- Warn, the default, enables you to decide on a case-by-case basis. When there is a perceived conflict
 between the two sources of execution control settings, a warning message appears. This enables you
 to resolve the conflict by right-clicking the Solution cell and selecting Edit Run Definition. You can
 then choose to use the execution control settings from either the Setup cell or the Solution cell for
 either this run or for all subsequent runs (until you change this Options setting).
- Use Setup Cell Execution Control cause the execution control specified by the Setup cell to be used.
 This is the equivalent of right-clicking the Solution cell and selecting Using execution control from Setup cell always.
- Use Solution Cell Execution Control cause the execution control specified by the Solution cell to be used. This is the equivalent of right-clicking the Solution cell and selecting Using execution control from Solution cell always.

Resolving Execution Control Conflicts

If you add or change Execution Control in ANSYS CFX-Pre in a way that conflicts with the Execution Control settings stored in the **Solution** cell, an error message appears when you attempt to update the **Solution** cell.

The **Using execution control from** <code>Setup/Solution</code> **cell** options enable you to decide how to resolve the conflict on a case-by-case basis. Alternatively, you can choose one of the **Using execution control from** <code>Setup/Solution</code> **cell always** options. The latter options change your Workbench Options for CFX. To reset that choice, go to **Tools** > **Options** > **CFX** and change the value of the **Set the default execution control conflict option for the Solution cell** field to one of:

Fluent

Specify the following options for the Fluent application. **General Options** are applicable to all new and pre-existing projects. **Launcher Options** are the default value for any new Fluent-based system that you create.

General Options

These options apply to all Fluent-based systems in the Workbench project, regardless of whether the system was created before, or after, the option is enabled. Note that these options are not saved with the project, and the settings are always applied to the currently loaded project.

Show Warning on Editing Setup if Solution Has Current or Initial Data

Allows you to determine whether a warning message should appear when solution data exists and you attempt to open ANSYS Fluent from the **Setup** cell. When you open Fluent from the **Setup** cell, the mesh and settings file associated with the **Setup** cell are loaded into Fluent which may or may not be what you want. When this option is selected, the warning dialog is shown. Default: Enabled.

Automatically Delete Old Solutions On Start Of New Calculation

Allows you to be able to automatically remove old solution data when starting new computation. Default: Enabled.

Default Options for New Fluent System

Once set, these options apply to all newly created Fluent-based systems in the Workbench project. They can be overridden by editing the properties specified for the **Setup** and **Solution** cell. The settings specified in the **Setup** and **Solution** cell properties are always respected and saved with the Workbench project. Therefore, for previously saved projects, these saved settings are used.

Launcher Options

These options apply to Fluent Launcher for new Fluent-based systems.

Show Launcher at Startup

Allows you to show or hide Fluent Launcher when Fluent starts. Default: Enabled.

Display Mesh After Reading

Allows you to show or hide the mesh after the mesh or case/data is read into Fluent. Default: Enabled.

Embed Graphics Windows

Allows you to embed the graphics windows in the Fluent application window, or to have them free-standing. Default: Enabled.

Use Workbench Color Scheme

Allows you to use either the classic black background color in the graphics window, or to use the Workbench color scheme. Default: Enabled.

Setup Compilation Environment for UDF

Allows you to specify compiler settings for compiling user-defined functions (UDFs) with Fluent. Default: Enabled.

Precision

Allows you to choose either the single-precision or the double-precision solver.

Setup Cell

The option applies to the **Setup** cell for new Fluent–based systems.

Enable Generation of Setup Output Case File

Allows you to bypass loading the mesh and setting files and reapplying the pre-set mesh operations every time the **Setup** cell is edited, resulting in faster runs. This option is especially beneficial when computing simulations across multiple design points involving ANSYS Fluent-related

parametric changes. The Fluent solver automatically generates the output case file, name-Setup-Output.cas, every time you

- modify the mesh or set mesh operations prior to running the simulation in ANSYS Fluent, or
- · start a Fluent session with a mesh file only

The generated output case file will be used when launching the next Fluent session from the **Setup** cell if the regular case file is out-of date or not available. Default: Enabled.

Solution Cell

These options apply to the **Solution** cell for new Fluent-based systems.

Enable Solution Monitoring

Allows you to be able to graphically view Fluent solution convergence and monitor data without having Fluent open. Once this option is enabled, you can use the **Show Solution Monitoring** option in the **Solution** cell context menu to display convergence and monitor charts. Default: Disabled.

Enable Generation of Interpolation File

Instructs the Fluent solver to automatically generate an interpolation file, *.ip, at the end of the run. Unlike *.dat file, the *.ip file can be used to restart the Fluent session, even if the mesh has been modified. Default: Disabled.

Mechanical

Specify the following options for the Mechanical application:

Pre-load the Mechanical editor Controls whether the Mechanical editor starts in the background when a Mechanical system is added to the Project Schematic or when a project containing a Mechanical editor system is resumed. Default: Disabled.

If you enable or disable this option, you must restart Workbench for your change to take effect.

Auto Detect Contact On Attach Controls whether contact detection is computed upon geometry import into Mechanical. Default: Enabled.

Release License for Pending Jobs Specifies when the Mechanical application is to release its license when running in batch mode:

Enable Legacy Solve Causes Mechanical to revert to an older method of launching the Mechanical APDL solver. This mode may have incompatibilities with newer features and will not affect jobs submitted to RSM. It is provided only as a diagnostic tool for troubleshooting.

Design Points Control periodic restarts of the Mechanical application when running in batch mode:

• During a design point update, periodically restart the Mechanical application Directs the Mechanical application to automatically restart after the specified number of design points when running in batch mode. Default: Enabled and set to restart after each (one) design point.

Each restart resets the Mechanical application and slightly lengthens the processing time, but can improve overall system performance (memory and CPU) when the generation steps of each design point (geometry, mesh, solve, post processing) are long. In such cases, specify a low number (minimum is 1) of design points before restarts.

In cases where the generation steps for each design point are short, reduce processing time by increasing the number of design points before restarts or prevent restarts completely by disabling this preference.

- On Demand When the Solution cell for a Mechanical system is in the pending state during a batch run, the right-click menu has a Release License option. This closes the Mechanical application but does not interfere with the completion of the run. (Default)
- **Always** Causes the Mechanical application to automatically close and release its license during batch runs when the **Solution** cell is in the pending state.
- **Design Point Run Only** Causes the Mechanical application to automatically close and release its license during **Update All Design Points** runs when the **Solution** cell is in the pending state.

Parallel Processing Controls the number of cores used for Data Mapping and Post-Processing.

Limit Number of Cores for Data Mapping and Post-Processing: Indicates the number of cores used by
the data transfer mapping, interpolation operations and result post-processing should be limited to a
user-specified value. The default is to use as many processors as available. If limited, the default is set to
two cores.

Microsoft Office Excel Options

Specify the following options for the Microsoft Office Excel add-in:

Named Ranges Filtering Prefix. If you wish to use a prefix to filter which Excel named ranges will be exposed as parameters in a Workbench project, enter that prefix here. All named ranges defined in the Excel file that include that prefix will be displayed as parameters in the project. By default, this setting is blank (no filter).

- If you specify a prefix, the prefix will be used to filter the named ranges for all new design exploration systems. For example, if you set this option to **WB**, the **Named Range Key** property (in the **Setup** section of the **Properties** view of the Microsoft Office Excel **Analysis** system) will be set to **WB** for all new projects you create.
- If you've specified a different filtering prefix the project level via the **Named Range Key** property, the project-level setting will not be affected by changes to this option.

Meshing

Specify the following options for the Meshing application.

Pre-load the Meshing editor Controls whether Meshing starts in the background when a Meshing system is added to the Project Schematic or when a project containing a Meshing system is resumed. Default: Disabled.

If you enable this option then later disable it, you must restart Workbench in order for your change to take effect.

Auto Detect Contact On Attach Controls whether contact detection is computed upon geometry import into Meshing. Default: Enabled.

Design Points Control periodic restarts of the Meshing application when running in batch mode:

• During a design point update, periodically restart the Meshing application Directs the Meshing application to automatically restart after the specified number of design points when running in batch mode. Default: Enabled and set to restart after each (one) design point.

Each restart resets the Meshing; application and slightly lengthens the processing time, but can improve overall system performance (memory and CPU) when the meshing of each design point is long. In such cases, specify a low number (minimum is one) of design points before restarts.

In cases where the meshing step for each design point is short, reduce processing time by increasing the number of design points before restarts or prevent restarts completely by disabling this preference.

Design Exploration Options

Specify the following options for the DesignXplorer application.

- Default options are the settings that will be used by default when new design exploration systems are created.
- General options are settings that are used for all design exploration systems. Once defined, they take effect immediately on all existing systems.

For a detailed explanation of default options for design exploration and local options for Design of Experiments, Response Surface, and Sampling and Optimization, see Design Exploration Options in the Design Exploration User's Guide.

Geometry Import

Specify your default geometry import options, including 2D/3D, types of bodies, material properties, geometry editor preferences, etc. You can also define advanced options, such as CAD associativity and import behavior.

Geometry Editor Options

Use SpaceClaim Direct Modeler as an External CAD Controls interactions between the Workbench and SpaceClaim Direct Modeler. When unchecked (default), SpaceClaim is fully integrated into the Project Schematic as a geometry editor. When checked, SpaceClaim is controlled as an External CAD system, with no direct integration into the Project Schematic.

Note

- Switching this preference from within an active project could result in project schematic inconsistencies and undesired behavior. It is advised that this preference be set during project creation and remain unchanged by all users interacting with the project.
- If the preference must be changed, the Geometry systems that reference SpaceClaim geometry should be **Reset** and reconfigured to properly release any previously stored parameters and file references.

 Preferred Geometry Editor Specifies the preferred geometry editor. You can choose either Design-Modeler (default) of SpaceClaim Direct Modeler.

Note

The SpaceClaim Direct Modeler option is not available when the **Use SpaceClaim Direct Modeler as an External CAD** option is checked.

CAD Integration Options

- CAD Licensing Specifies if the plug-in license should be retained or released after the CAD import/refresh.
 - Hold: Instructs ANSYS Workbench to keep the license after the import or refresh operation has completed.
 The option is useful when executing design studies.
 - Release: Instructs ANSYS Workbench to free the license for someone else's use after the import or refresh
 operation has completed.

Note

If the **Hold** option was previously set and subsequently changed to **Release**, the license will be released immediately after clicking **OK** on the **Options** dialog box. A plug-in license that is held will always be released when exiting the ANSYS Workbench session.

Analysis Options

Analysis Type Specifies the analysis type. You can choose either 3D or 2D. For more information, see
the Advanced Geometry Options table at Geometry Preferences in the CAD Integration section of the
ANSYS Help.

Basic Options

For detailed descriptions of the basic geometry import options, see the **Basic Geometry Options** table at Geometry Preferences in the CAD Integration section of the ANSYS Help.

Advanced Options

For detailed descriptions of the advanced geometry import options, see the **Advanced Geometry Options** table at Geometry Preferences in the CAD Integration section of the ANSYS Help.

STL Specific Options

The STL Specific options are applicable to beta features and therefore are not supported.

Using Software Licensing in ANSYS Workbench

ANSYS Workbench offers two licensing methods at Release 15.0:

• Share a single license between applications (default) (shared mode)

Use a separate license for each application (separate mode)

Use the Licensing Preferences dialog box (**Start> All Programs > ANSYS 15.0 > ANSYS Client Licensing > User License Preferences 15.0**) to specify which method to use and which licenses to use. You must specify the licensing method before starting an ANSYS Workbench session. If you access the Licensing Preferences dialog box from the ANSYS Workbench Tools menu, you will not be able to choose a licensing preference from there.

Single License Sharing

ANSYS Workbench allows you to work across multiple applications and workspaces in ANSYS Workbench while consuming only one of a single type of license per user per session. Using shared licensing, the active application holds the license, preventing other applications that are sharing that license from using it during that time. The application or operation requiring use of the license is called a concurrency event. For example, meshing and solving would each be a concurrency event.

Single license sharing allows you to progress through your analysis, from specifying engineering data through building or attaching a geometry, meshing, setup, solving, and finally, reviewing your results, all under the same license. The application holding the license must close or issue a PAUSE command, or receive an automatic release request to release the license and allow another application to use it. Licenses cannot be released while an application is actively performing a licensed operation (for example, an application cannot release a license in the middle of a solve operation; the license cannot be released until the solve operation is completed).

Single license sharing applies only to licenses of the same type (e.g., Mechanical). Choosing this option does not affect your ability to use licenses of different types simultaneously (e.g., Mechanical for one task and Fluid Dynamics for another).

Because this method is the default, you do not have to take any action to run this way.

Explanation of License Type and Examples License type is primarily by license feature. It is possible to use both a Mechanical and an Emag license within a single ANSYS Workbench session. It is also possible to use both a Multiphysics and a Mechanical license within a single ANSYS Workbench session.

The first license checked out within a session will be based on your preferences and what capabilities are being requested. For all applications other than the first (subsequent) one opened (within ANSYS Workbench), ANSYS licensing will first look at what other licenses are opened within this session. These subsequent license requests will look at sharing first to satisfy their request: do any other licenses being used within this session fulfill the needed capabilities? If yes, share an existing license. If not, preferences are used and a new, different license is checked out.

Example 1: You have one license for Multiphysics and one license for Mechanical, with Multiphysics listed first in your preferences. The first application starts and only needs capabilities in Mechanical. Since Multiphysics contains Mechanical capabilities and is first in your preferences, Multiphysics will be checked out. The second application starts and needs Multiphysics; since Multiphysics is already checked out, the second application will share it with the first. Only the Multiphysics license is consumed in this session.

Example 2: You have one license for Multiphysics and one license for Mechanical, with Mechanical listed first in your preferences. The first application starts and only needs capabilities provided in Mechanical, so Mechanical is checked out. The second application starts and needs capabilities provided on Multiphysics; since (the already in use) Mechanical cannot satisfy its requirements, it checks out Multiphysics. Both a Multiphysics and a Mechanical license are consumed in this session.

Restrictions of Single License Sharing You cannot run two concurrency events simultaneously (for example, you cannot mesh one model and solve another simultaneously) with one license.

If you are using a license for one application, other applications may still not be able to share that license if those applications require capabilities not supported by the license. For example, you cannot share a Mechanical license with a Fluent application.

Single License Sharing in ANSYS Workbench Applications ANSYS Workbench applications handle single license sharing differently:

The Mechanical Application:

You can launch the Mechanical application and move between its components (such as Meshing, Setup, and Solve). The active component will control the license while completing its operations and will release the license as soon as the operation is completed. For example, when you mesh, the Meshing component will control the license during the meshing operation and then immediately release the license when the operation is completed. The other components will remain in a read-only mode while Meshing uses the license, allowing you to view the data in other components but not operate on it.

Note

Applications in read-only mode because of shared licensing do not refresh their license status automatically. Once the shared license is released by the editor that had consumed it, you must trigger Mechanical to query the license status. The most straightforward way to do this is click outside the Mechanical application window and then click back in the window to cause the license availability to be rechecked.

The Mechanical APDL Application:

This application consumes a license as soon as you launch it, and retains that license until it is finished. If you launch the Mechanical APDL application interactively, the license is retained until you either close the application or issue a PAUSE command at the Mechanical APDL command line. PAUSE allows you to temporarily release the license for another application to use. No other operation other than SAVE or /EXIT is permitted while PAUSED. When the second application has finished and releases the license, issue an UNPAUSE command from the Mechanical APDL command line to resume its use of the license.

CFX, Fluent, Autodyn, Polyflow:

These applications consume a license when launched and retain the license until they receive a request from another application to release it. For example, if you open CFX-Pre, CFX-Pre will obtain and control the license. It will retain the license until you close the application or until another application (such as the CFX solver) requests it.

Autodyn and Polyflow also provide a manual PAUSE feature that allows you to interrupt Autodyn or Polyflow and release the license, temporarily, for another application to use.

Separate Licenses

By using the separate licenses method, ANSYS Workbench requires a separate license for each application. By using this method, you can move freely between the many applications that you might require during an analysis in ANSYS Workbench, provided that you have sufficient licenses. You can leave each application running and easily move between them at any point during the analysis, even if one of the applications is actively using the license (such as during a solve process). The disadvantage to this method is that you could potentially consume many licenses.

To activate the separate licenses method, choose **Use a separate license for each application** in the Licensing Preferences dialog box (**Start> All Programs > ANSYS 15.0 > ANSYS Client Licensing > User License Preferences 15.0**). You must specify the licensing method before starting an ANSYS Workbench session.

Examples of Using Separate Licenses You have two Mechanical licenses. When you open and mesh or solve a model in the Mechanical application, you consume one Mechanical license. If you link that Mechanical analysis to a Mechanical APDL system, you would consume a second Mechanical license when you launch the Mechanical APDL application, if you have not closed out of the Mechanical application. Neither of these licenses would then be available for other users until you closed out of one or both of the applications.

Explicit Product Licensing

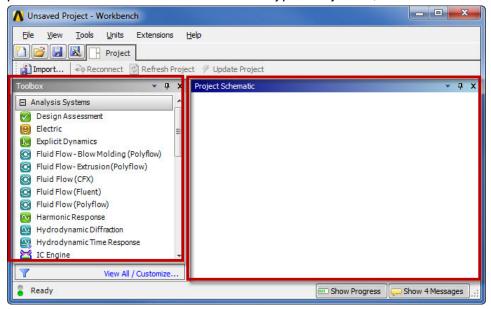
Licenses that can be used to start the Mechanical application products are called primary configured tasks. Licenses that cannot start a product but add functionality are called add-on licenses. Most licenses are either one or the other. A primary configured task license will only be pulled during the launch of the application and cannot be added on-demand if the application is already open. For example, the Multi-Body Dynamics analysis is only supported as an add-on license, so another, primary configured task license will be needed to start the Mechanical application for a Transient Structural (Rigid dynamics) analysis. An add-on license will be used whenever the need arises, even if the Mechanical application is opened. However, the ANSYS Explicit STR products like AUTODYN-2D, AUTODYN-3D, and ANSYS Explicit STR are of a dual nature and are supported both as primary configured tasks and as add-ons. The dual nature of explicit products enables the analysis of the mixed implicit and explicit system using a single Mechanical editor.

Release 15.0 - © SAS IP, Inc. Al	Trights reserved - Conta	inc proprietary and	confidential inf	ormation
nelease 13.0 - @ 3A3 IF, IIIC. AI	i ngnis reserveu Contu	iiris proprietary ana	Commuentiarini	omination
-11	NSYS. Inc. and its subsid	l:		
OI A	an sy s. inc. ana its subsia	iaries ana attiliates.		

Working in the ANSYS Workbench Project Tab

In the **Project** tab, you will take systems from the **Toolbox** and add them to the **Project Schematic**. Systems are added from left-to-right, and from top-to-bottom. All data transfer occurs from left (also called upstream) to right (also called downstream); you cannot transfer data from right to left. Therefore, when placing or moving systems, it is important that you place receiving systems to the right of sending systems. All processing of data (updates, etc.) also occurs in the same direction, from left-to-right and top-to-bottom. Again, be aware of this order when placing or moving systems.

Most analysis systems are defined by three primary attributes: physics type, analysis type, and solver type. ANSYS Workbench uses these attributes to determine valid data transfer and system replacement possibilities. For more information on the types of systems, see ANSYS Workbench Systems (p. 135).



Related Topics:

Adding Systems to the Project Schematic Naming and Renaming Systems Working through a System Creating and Linking a Second System Duplicating Systems Moving, Deleting, and Replacing Systems

Adding Systems to the Project Schematic

The first step in building an analysis is to take one or more systems from the **Toolbox** and add them to the **Project Schematic**. ANSYS Workbench offers the following methods of adding new systems:

- double-click the system in the Toolbox
- drag the system from the Toolbox and drop it into the Project Schematic

• right-click the **Project Schematic** and select the system from the context menu

Adding a System by Double-Clicking

The simplest way to add a new system to the **Project Schematic** is to double-click on the desired system in the **Toolbox**. (A single-click simply selects an object and does not modify data or initiate any action.) The system will be placed in a new row in the **Project Schematic**, below any existing systems.

Adding a System using Drag-and-Drop

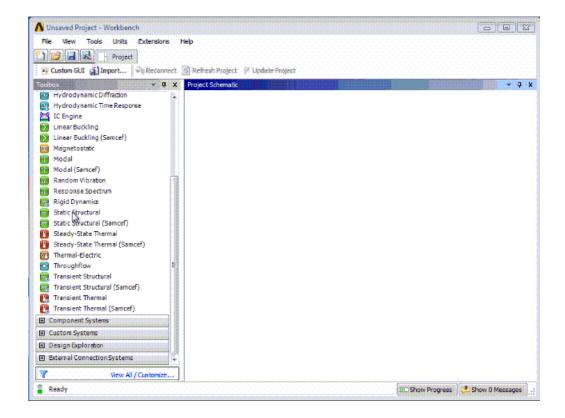
You can add a system to the **Project Schematic** by dragging it from the **Toolbox** and dropping it in the desired location. This method provides a preview of possible target locations, allowing you to choose the best location for that system. The green boxes in the following animation indicate possible drop targets.

If you move the mouse over one of the drop targets, the box will change to red and text will indicate the result of dropping at that location.

In cases where the new system can be linked to one or more existing systems, drop targets are also shown on the eligible cells of the existing system(s).

The following animation demonstrates adding systems via the double-click and drag-and-drop methods. In this example, several cells of the Structural Static system are also possible drop targets of the Modal system. For more information, see Creating and Linking a Second System (p. 43).

The following Show-Me Animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.

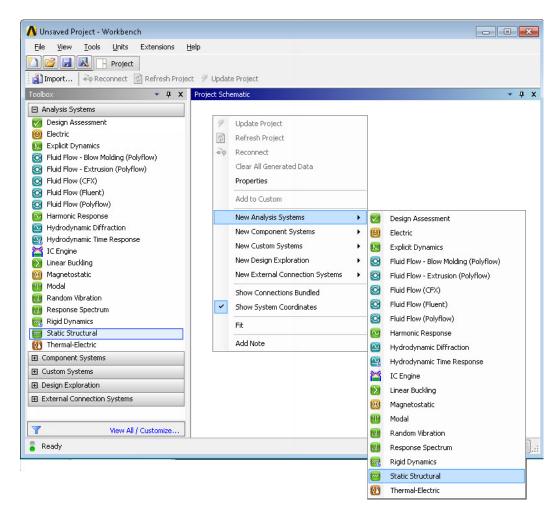


Note

When you attempt to add an invalid system (for example, if the mesh type is not compatible with the system you are attempting to add), the drop target preview will be visible, but the system will not be added when you release the mouse. Details are written to the Messages View (p. 251).

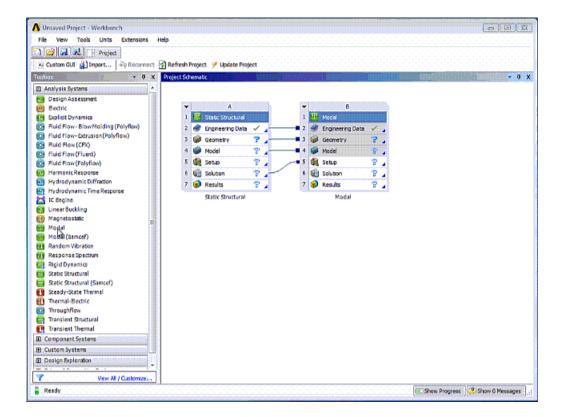
Adding a System using the Context Menu

You can add a system to the **Project Schematic** by right-clicking on white space in the **Project Schematic**. The context menu includes a list of **New ... Systems** options, allowing you to select the system category and a specific system within that category (the system options reflect your **Toolbox** configuration). For example, to introduce a new Static Structural system, right-click on **Project Schematic** white space and select **New Analysis Systems > Static Structural**, as shown below.



To add a new system with a link to an existing system, right-click on the appropriate cell of the existing system. Select **Transfer Data From New** to create a new system upstream of the selected cell. Select **Transfer Data To New** to create a system downstream from the selected cell. The following animation demonstrates adding a system using the **Transfer Data** context menu options.

The following Show-Me Animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



For more information, see Creating and Linking a Second System (p. 43).

Naming and Renaming Systems

In general, it is good practice to give each system a descriptive name that is meaningful for you and that indicates the details of the system. ANSYS Workbench allows you specify a name for each system, either initially when the system is added to your project, or at any time afterward.

Naming Systems

When a new system is added to the **Project Schematic**, the default name of the system will be in focus (highlighted), as shown below.



When the name is in focus, it is editable and you can enter a new name for the system. To accept the default name for the system (the default name is usually the same as the system type), click **Enter** or select any other action in the user interface. In the example shown below, we have entered the name "My Structural Analysis" and clicked **Enter**.



After you click **Enter** (either after entering a new name or accepting the default name), the focus moves to the system cell requiring attention first (see Understanding Cell States (p. 255)). In the example above, this cell is the **Geometry** cell. By applying focus in this manner, ANSYS Workbench draws your attention to the cell where you will most likely begin working with your system (see Working through a System (p. 38)).

Renaming Systems

To rename an existing system, you can double-click on the system name. Alternatively, you can right-click the system header (Row 1) and select **Rename** from the context menu. The focus will move to the system name, as shown below. Type the new name and click **Enter**.



The example below shows the system renamed to "My Structural Analysis."



Working through a System

ANSYS Workbench provides you with a fairly straightforward workflow for creating and working through a system. First, you select a system from the **Toolbox** and add it to the **Project Schematic** (see Adding Systems to the Project Schematic (p. 33)). Then you work through the cells in the system, generally from top-to-bottom, until you have completed all the required steps for your analysis. In most cases, data flows from top-to-bottom through the system, as well. For example, in a Mechanical system, the geometry must be defined before you can define the model; the **Model** cell uses the geometry defined in the **Geometry** cell as its input.

Because the workflows for different types of analyses differ to some degree, we have included two typical examples of working with analysis systems: one for a mechanical analysis (see Basic Mechanical Analysis Workflow (p. 40)) and one for a fluid flow analysis (see Basic Fluid Flow Analysis Workflow (p. 41)).

Defining your Simulation Geometry

All analysis systems and several component systems, including Geometry, Meshing, and Mechanical Model, begin with a geometry-definition step. You can define the geometry differently depending on the type of simulation you are running. In most cases, you will use the **Geometry** cell. Via the **Geometry** cell, you can:

- · Create a geometry from scratch in DesignModeler
- Import an existing geometry:
 - From neutral formats like IGES, STEP, Parasolid, ACIS
 - From CAD files on disk
 - From an active CAD session that is already running on the same machine

For Fluid Flow simulations, you can also start with an imported mesh or case file; see Basic Fluid Flow Analysis, Starting from an Imported Mesh (p. 42) for details.

Specifying Geometry via the Context Menu

- 1. Right-click the **Geometry** cell.
- 2. Choose **New Geometry** to launch DesignModeler and create a new model, or choose **Import Geometry** and browse to an existing CAD model.

Alternatively, you can also launch ANSYS Workbench directly from some CAD systems. When doing so, ANSYS Workbench starts with a Geometry system in place and the CAD file already attached.

After you have attached or imported your geometry, the state appears as Up to Date, and the icon indicates the type of file imported. Geometry types include:

```
ACIS (.sat)

ANSYS Neutral File (.anf)

Autodesk Inventor (.ipt, .iam)

BladeGen (.bgd)

CATIA v4 (.model, .dlv)

CATIA v5 (.CATPart, .CATProduct)
```

Creo Elements/Direct Modeling (.pkg, .bdl, .ses, .sda, .sdp, .sdac, .sdpc)



If you do not need to make any additional changes to your geometry, you can continue working through the analysis as described in the next sections.

If your geometry needs to be modified before continuing with your analysis, you can edit the geometry in DesignModeler. After modifying the geometry in DesignModeler or importing a DesignModeler file, the icon in the **Geometry** cell will change to a DesignModeler icon. For a file imported and then modified in DesignModeler, you can open the file in DesignModeler, and the DesignModeler model tree will indicate the original source of geometry.

After the geometry is defined, you can share it with other systems. See Data Sharing and Data Transfer for more information on sharing geometry systems.

For detailed CAD-related information specific to the ANSYS DesignModeler application and ANSYS Workbench, see the CAD Integration section of the product help. When accessing the **ANSYS Workbench Help** from the **Help** menu, click the **Contents** tab and open the **CAD Integration** folder in the hierarchical tree to access the CAD Integration section.

Basic Mechanical Analysis Workflow

After introducing a new Mechanical analysis system to the **Project Schematic** and assigning an appropriate name, the focus is typically directed to the **Geometry** cell, because this is usually the first cell in the system that requires user input. You could edit or define material models via the **Engineering Data** cell, but this example assumes that the default materials will suffice.

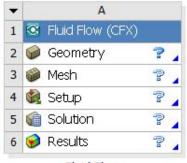


As noted earlier, you typically work through the system from top to bottom. Use the context menus for each cell to view and select operations that can be performed for that cell.

- Attach geometry to your system or build a new geometry in DesignModeler. Right-click the Geometry
 cell and select Import Geometry to attach an existing model or select New Geometry to launch
 DesignModeler.
 - For details, see Defining your Simulation Geometry (p. 39).
- Define all loads and boundary conditions. Right-click the **Setup** cell and select **Edit**. Mechanical will
 open. Set up your analysis using that application's tools and features. For more information on setting
 up and running specific Mechanical analyses, see Analysis Systems (p. 135).

Basic Fluid Flow Analysis Workflow

After introducing a new Fluid Flow analysis system to the project schematic and assigning an appropriate name, the focus is typically directed to the **Geometry** cell, because this is usually the first cell in the system that requires user input. An example of both a CFX and a Fluent fluid flow system is shown below.





Fluid Flow Fluid Flow (FLUENT)

As noted earlier, you typically work through the system from top to bottom. Use the context menus for each cell to view and select operations that can be performed for that cell. For fluid flow systems, the process is somewhat flexible; you can start from geometry, from an existing mesh, or from an existing case file; each is described in the following sections.

Basic Fluid Flow Analysis, Starting from Geometry

1. Attach geometry to your system or build a new geometry in DesignModeler. Right-click the **Geometry** cell and select **Import Geometry** to attach an existing model or select **New Geometry** to launch DesignModeler.

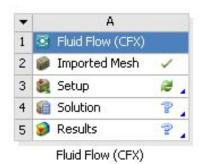
For details, see Defining your Simulation Geometry (p. 39).

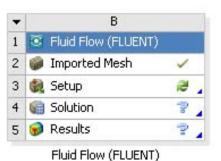
- 2. After your geometry is defined and the **Geometry** cell shows the green check mark indicating that the cell is up to date, you can proceed to the **Mesh** cell. Double-click on the **Mesh** cell or right-click and select **Edit** to launch the Meshing application. Note that if you want to generate a default mesh, you could also right-click on the **Mesh** cell and selected **Update** to generate the mesh in the background without launching the meshing application. When this step has successfully completed, the Mesh cell shows a green check mark indicating an up-to-date state.
- 3. Double-click on the **Setup** cell or right-click and select **Edit** to load the physics pre-processor. If you are working a Fluid Flow (Polyflow) system, the editor will be the Fluent application. If you are working with a Fluid Flow (ANSYS CFX) system, the editor will be CFX-Pre. When you have successfully defined your physics, the **Setup** cell shows a green check mark indicating an up-to-date state.
- 4. Double-click on the **Solution** cell or right-click and select **Edit** to open the associated **Solution** cell editor. If you are working a Fluid Flow (Polyflow) system, the editor will be the Fluent application. If you are working with a Fluid Flow (ANSYS CFX) system, the associated editor will be the CFX-Solver Manager. You can also right-click and select **Update** to run the solution in the background. When the solution is complete, the Solution cell shows a green check mark indicating an up-to-date state.
- 5. You can now post-process the resulting solution. Double-click on the **Results** cell or right-click and select **Edit** to open the resulting solution in ANSYS CFD-Post.
- 6. Once the process has been completed, all cells in the system should show a green check mark indicating an up-to-date state.

Basic Fluid Flow Analysis, Starting from an Imported Mesh

It is also possible to bypass the **Geometry** cell, and begin the simulation process by importing a mesh.

1. Right-click on the Mesh cell and select Import Mesh File. In the Open dialog, browse to find your desired mesh file and select Open. The process for CFX and Fluent differs only in the types of files that are supported (as shown by the options shown in the Files of Type drop-down list). After selecting Open, the Mesh cell will change to Imported Mesh, and will show a green check mark to indicate an up-to-date state. The Geometry cell (which must be unused for the Import Mesh File option to be shown in the Mesh cell context menu) will be deleted from the system (examples shown below for both a CFX and a Fluent-based Fluid Flow system).





2. Double-click on the **Setup** cell or right-click and select **Edit** to load the physics pre-processor. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 41).

Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (ANSYS CFX) System

It is also possible to bypass both the **Geometry** cell and the **Mesh** cell and begin the simulation process by importing a case file. Here the process differs slightly for CFX and Fluent-based systems; each is described below.

- 1. Right-click on the **Setup** cell and select **Import Case** > **Browse**. In the **Open** dialog, browse to find your desired case file and select **Open**.
- 2. If the **Geometry** and **Mesh** cells are unused (empty), then the unused cells are automatically deleted. Note that if either cell has an incoming or outgoing connection, then it is considered used, and will not be deleted.
- 3. Since CFX supports multiple meshes imported into a single setup cell, if either of the **Geometry** or **Mesh** cells contains data, both cells will not be deleted. As a result, both the Mesh generated in the **Mesh** cell and the mesh imported from the case file will be combined in CFX-Pre.
- 4. Once the import is complete, you can double-click **Setup** cell or right-click and select **Edit** to start CFX-Pre. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 41).

Basic Fluid Flow Analysis, Starting from an Imported Case File: Fluid Flow (Polyflow) System

- 1. Right-click on the **Setup** cell and select **Import Polyflow Case**. Choose either from the list of recently used case files or choose **Browse...**.
- 2. A warning dialog informs you that completing this action will result in the deletion of the **Geometry** and **Mesh** cells and any associated data. Click **OK**.
- 3. After selecting the desired case file, the **Geometry** and **Mesh** cells will be deleted from the system. Any existing connections to the **Geometry** and **Mesh** cells will also be deleted. From this point, follow the procedure described in Basic Fluid Flow Analysis, Starting from Geometry (p. 41).

Creating and Linking a Second System

After you have created the first system in the **Project Schematic**, you can create additional systems. A new system can be either an independent system or a connected system that has data in common with other systems.

Related topics:

Creating Independent Systems Creating Connected Systems

Creating Independent Systems

Independent systems are systems on the **Project Schematic** that are not connected to other systems via data links. Multiple independent systems can be used to keep the analyses for related but separate components within the same project (for example, two parts of the same vehicle). You may also want to create independent systems if you are analyzing the same model but using different solvers to compare the results, or are using different editors on independent yet related data (for example, you

are using Mechanical APDL on an independent input file, and using Mechanical on a geometry file that has related origins to the input file of the Mechanical APDL system).

You can create a new independent system via either of the following methods:

- Double-click the desired system in the **Toolbox**. The new system will be created on the **Project Schematic** below and will not be linked to any existing systems.
- Use a drag-and-drop operation. When you drag the system from the **Toolbox** and move it over the **Project Schematic**, you will see all a preview of all possible drop targets, including the possible links for each target. Be sure to drop the template on the target that best reflects your engineering intent (which in this case, is to create an independent system).

For an animated example, see Adding a System using Drag-and-Drop (p. 34).

Creating Connected Systems

Connected systems are systems on the **Project Schematic** that are connected together via data links so data can be shared and/or transferred between the two systems. You can use connected systems for sequential physics coupling (e.g., thermal-stress) or for sequential simulation steps, such as a prestress modal (stress analysis followed by modal), mode superposition (modal followed by harmonic), etc. You can either create connections between existing systems or create a new system that is connected to an existing system.

Note

When using a drag-and-drop operation to create connections, keep in mind that when multiple drop targets are available, each one results in a different set of connections.

Connecting Two Existing Systems

To connect two existing systems, use a drag-and-drop operation to share one or more component cells (for example, a **Geometry** cell and/or an **Engineering Data** cell) from one system with eligible cells in another system. To preview possible drop targets, drag a cell from the source system and hold it over cells in the target system. Drop the system on the target best suited to your engineering goals. In this case, the data is shared and the two systems are then independent of each other.

Creating a New Connected System

When you create a new system that is connected to an existing system, ANSYS Workbench generates shares and/or data transfers between the systems to achieve a compound analysis. In this case, the data is shared and the two systems are then interdependent.

The new system can be created either upstream (i.e., the new system provides inputs to the existing system) or downstream (i.e., the new system receives inputs from the new system) of the existing system. The system receiving input is also called the *dependent* system. To create a new connected system, you can use either a drag-and-drop operation or ANSYS Workbench's **Transfer Data** context options.

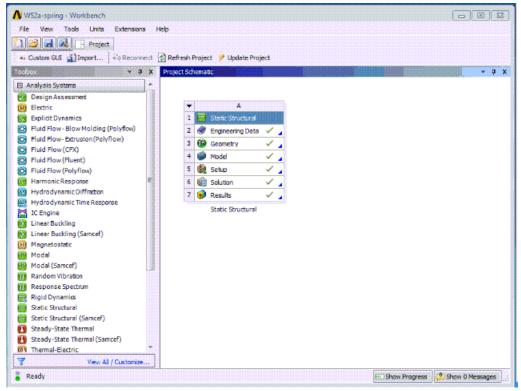
Use Drag-and-Drop

If you already know what kind of system you want to create, you can use drag-and-drop functionality to create a new dependent system. Drag a system from the **Toolbox**, move it over cells in the existing system to preview available drop targets, and drop the system on the desired target. The new system will be created downstream on the **Project Schematic**.

The following animation demonstrates the various drop targets and their results. For example, selecting the **Geometry** cell as the drop target will result in a second system that will share the Engineering Data and Geometry data. Selecting the **Model** cell as the drop target will result in a second system that will share the Engineering Data, Geometry and Model data. The preview for each drop target will indicate how the new system will be connected to the existing one.

If you select the **Solution** cell as the drop target, the preview will show the text "Share A2:A4 Transfer A6," meaning that the data for cells A2, A3, and A4 (**Engineering Data**, **Geometry**, and **Model** in the following animation) would be shared, and data from cell A6 (in this case, **Solution**) would be transferred as input to the new system. It is important to review each potential drop target carefully to ensure that you select the target that best suits your needs. Note that in addition to the red, linked drop targets, you can also preview the green independent drop targets.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



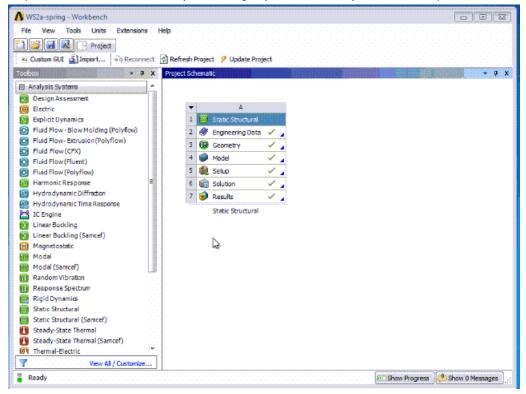
Use Transfer Data Context Options

The **Transfer Data** options available in the context menu allow you to select a cell in an existing system and create a new system either upstream or downstream of that system.

- To create a new system upstream of the existing system (so the existing system is dependent), rightclick the target cell in the existing system and select **Transfer Data from New**. Select your new system from a list of all possible analysis types that can *provide data to* the existing cell.
- To create a new system downstream of the existing system (so the new system is dependent), rightclick the target cell in the existing system and select **Transfer Data to New**. Select your new system from a list of all possible analysis types that can *accept data from* the existing cell.

When using either of the **Transfer Data** options, all possible cells will be shared, up to the position of the selected cell. The following animation demonstrates using the **Transfer Data to New** option to add an upstream Linear Buckling system to an existing Static Structural system.

The following Show-Me animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



Duplicating Systems

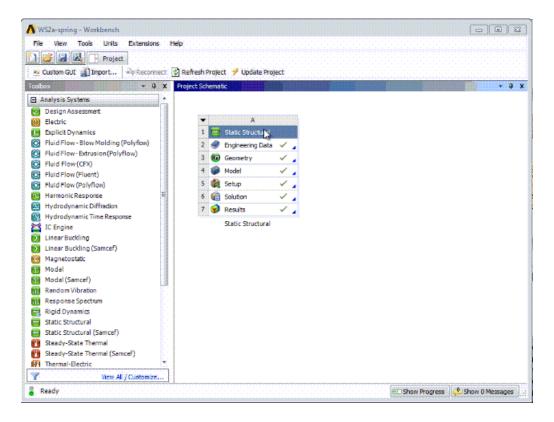
To duplicate an existing system, right-click it and select **Duplicate** from the context menu. The result of the operation will depend on the cell from which the Duplicate operation is initiated.

To create a duplicate system in which all cells can be edited independently of the original system, right-click the system header and select **Duplicate**, as shown in the image below.

Note

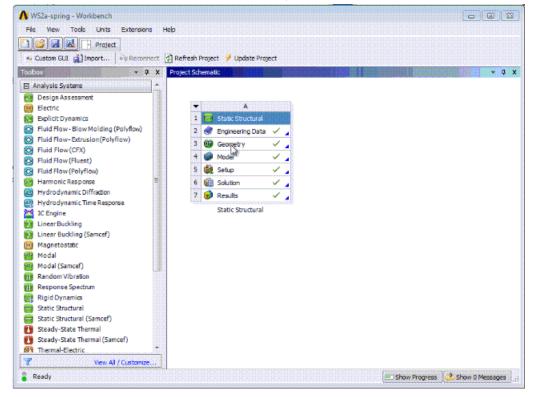
When a Mechanical system containing a **Results** cell is duplicated, the results will NOT be copied to the new system.

The following Show-Me Animations are presented as animated GIFs in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



If the Duplicate operation is initiated from one of the cells in the system, all cells above the one selected for duplication will be shared. The cell selected for duplication can be edited independently. For example:

• If you select **Duplicate** from the **Geometry** cell, the **Engineering Data** cell is shared, allowing you to edit the **Geometry** cell in the duplicate system to investigate a geometric alternative. Data from the **Model** cell and below is copied from the original system and can be modified independently.



- If you select **Duplicate** from the **Model** cell, the **Engineering Data** and **Geometry** cells are shared, allowing you to edit the **Model** cell in the duplicate system to investigate an alternative modeling approach. All data in the **Setup** cell and below is copied from the original system and can be modified independently.
- If you select **Duplicate** from the **Setup** cell, the **Engineering Data**, **Geometry** and **Model** cells are shared, allowing you to edit the **Setup** cell in the duplicate system to investigate alternate loads and constraints. All data in the cells below **Setup** is copied from the original system and can be modified independently.

To duplicate multiple, connected systems, (the equivalent of duplicating at the Model level with multiple environments in previous releases of the Mechanical application), you must use the **Export** capability in the Mechanical application to save a .mechdat.Then use ANSYS Workbench's Importing Legacy Databases (p. 78) capability to import the .mechdat into your project to create the duplicated set of systems.

Note

When a Mechanical system containing a **Results** cell is duplicated, the results will NOT be copied to the new system.

Moving, Deleting, and Replacing Systems

Moving a System

You can move an existing system to another position on the **Project Schematic**. To move a system, click on the header cell and drag the system to the new location. The preview will indicate possible target locations for the system.

Deleting a System

To delete a system from the **Project Schematic**, right-click the system header cell and select **Delete**.

Replacing a System

To replace an existing system with different type of system, right-click the system header cell and select **Replace With**, and select the type of system that will replace the existing system. The context menu contains a list of all system types that are eligible to replace the existing system.

Note

The units setting specified in an existing system is not maintained in the replacement system. In the replacement system, you must specify the units setting you want to use.

Working in ANSYS Workbench

In this section, we'll address working in ANSYS Workbench after you've populated your **Project Schematic** with systems.

The following topics are covered:
Using Remote Solve Manager (RSM) in ANSYS Workbench
Using Journals and Scripts
Project File Management
Working with the Chart View
Working with Project Reports
Using Help
Troubleshooting

Using Remote Solve Manager (RSM) in ANSYS Workbench

The following sections address using Remote Solve Manager in ANSYS Workbench.
Submitting Projects to Remote Solve Manager (RSM) for Remote Update
Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes

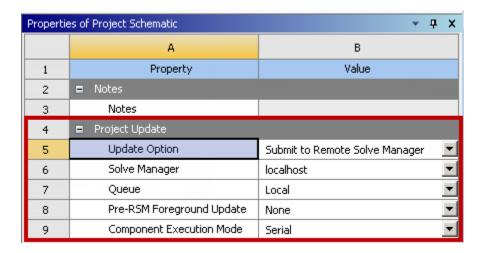
Submitting Projects to Remote Solve Manager (RSM) for Remote Update

You can submit a project to the Remote Solve Manager (RSM) for remote update of the full project. To do so, you must first configure the project properties and then use the **Update Project** option to submit the update.

Note

If you are sending solve jobs to a remote computing cluster, note that the license preferences set for your local machine (the RSM Client) may not be the same as the license preferences set for the remote cluster machine (the Compute Server). In this case, the Compute Server license preferences will be used for all jobs. For more information, see Establishing User Licensing Preferences.

- 1. Access project properties by right-clicking in the white space of the **Project Schematic** and selecting the **Properties** content menu option. Alternatively you can select the **View > Properties** menu option when the **Project Schematic** is open.
- 2. Under the **Project Update** category, set the **Update Option** property to **Submit to Remote Solve Manager**.



- 3. Additional RSM-related properties that are displayed. Use these properties to specify your remote submission settings.
 - **Solve Manager**: Specify the machine configured to manage the queues and compute servers to be used for the remote solution. Any Solve Managers currently defined for RSM will appear in the dropdown list. Select an available Solve Manager or specify the name of another machine.
 - Queue: Select from the queues already defined for RSM.
 - Pre-RSM Foreground Update: Select None to specify that you want to do a local geometry-only update
 prior to submitting project updates to RSM. Select Geometry to update your geometry locally before
 submitting project updates to RSM.

Component Execution Mode: Specify serial or parallel solver execution mode. The parallel option is available only if the selected solver supports parallel execution mode. This option may not be available with all systems. When performing a design point update via RSM with component update in the foreground, the **Parameter Set** properties will override the parallel-process settings in individual components. For details on updating design points, see Updating Design Points via Remote Solve Manager (RSM) (p. 107).

The **Project Update** properties specified here are shared with the **Parameter Set** as **Design Point Update Process** properties. Changes to the values of these properties here will be reflected in the **Parameter Set** properties, and vice versa.

Note

In most cases, the **Update Project** button or menu item updates all systems and cells in a project. When a full-project update is submitted to RSM as described, however, only those systems above the **Parameter Set** bar are sent to RSM for remote update. If needed, DesignXplorer systems can be further updated once the remote project update is completed.

Submitting Solutions for Local, Background, and Remote Solve Manager (RSM) Processes

You can use the **Solution Process** properties on the **Solution** cell of systems in your project to control where you run the solution (update) process. You can choose from the update options listed here. The

defaults for these options are determined by the settings specified in the **Solution Process** preferences under **Tools> Options**.

Note

If you are sending solve jobs to a remote computing cluster, note that the license preferences set for your local machine (the RSM Client) may not be the same as the license preferences set for the remote cluster machine (the Compute Server). In this case, the Compute Server license preferences will be used for all jobs. For more information, see Establishing User Licensing Preferences.

• **Run in Foreground** -- the default manner of running, where solutions are run within the current ANSYS Workbench session. This option is appropriate for quick-running solutions that fit within the resources of your workstation. This option is also the most robust as it is not possible to make changes that impact the current solution. When a solution is executing in the foreground, you cannot change or save the project, but you can interrupt or stop the solution.

Note

For Polyflow and CFX component or analysis systems, you can switch an update in progress on the local machine to background mode by right-clicking the **Solution** cell and selecting **Switch Active Solution to Background**.

• Run in Background -- allows you to run the solution in the background on the local machine. This option is appropriate for solutions that fit within the resources of your workstation but will take longer to execute. When a solution runs in the background, the cell enters the Pending state and you can interact with the project to exit ANSYS Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

Note

If you choose the **Run in Background** option for multiple solution components or for one solution component with multiple design points, your project and design point updates will run multiple instances of the solver at the same time. If you are using the **Share single license between applications when possible** option in the license preferences, only one of the solver runs will succeed. The others will fail because they cannot access the single license. There are two workarounds for this problem:

- Change the license preference to be Use a separate license for each application. This
 preference will allocate one license for each running solver.
- Instead of using the Run in Background option, use the Submit to Remote Solve Manager option and select the localhost as the Solve Manager and use the Local queue. RSM will

queue the solve jobs so that only one is running at a time, and only one solver license will be in use at a time.

Note

For Mechanical APDL, the **Download Distributed Files** property is not available and will not appear in the **General Property** table during foreground and background updates.

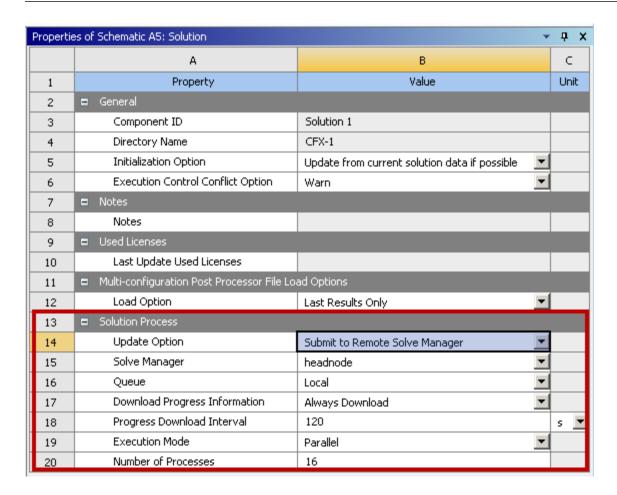
- Use application default -- (Mechanical application only). Uses the solver settings specified in the Mechanical application.
- **Submit to Remote Solve Manager** -- allows you to run the solution in the background by submitting the solution to Remote Solve Manager (RSM). This option is used primarily for long running solutions that do not fit within your workstation's resources. Through submission to RSM, the solution can be executed on remote computing resources. RSM can also submit jobs to the local machine to allow the queuing of solutions on your workstation. When a solution is submitted to RSM, the **Solution** cell enters the Pending state, similar to the **Run in Background** option.

Important

When design points are configured to be updated via RSM, the **Solution** cell cannot also be updated via RSM. If you have configured design points to be updated via RSM, change the **Solution** cell update settings by setting the **Update Option** Solution Process property to **Run in Foreground**. For special circumstances, however, the update of both the **Solution** cell and design points via RSM can be enabled. For assistance with enabling this functionality and configuring your system to support, go to the Support page of the ANSYS Customer Portal and submit an online support request. For further information about tutorials and documentation on the ANSYS Customer Portal, go to http://support.ansys.com/docinfo.

To use the Remote Solve Manager options, you must first configure RSM. For tutorials featuring step-by-step instructions on configuring RSM, go to the Downloads page of the ANSYS Customer Portal. For further information about tutorials and documentation on the ANSYS Customer Portal, go to http://support.ansys.com/docinfo.

If you choose **Submit to Remote Solve Manager**, you also have the following options, which are populated from or must use the settings from your existing RSM configuration.



- Solve Process Setting -- (Mechanical application only). This setting is the solving configuration that you
 have defined in Mechanical.
- **Solve Manager** -- This is the machine that is configured to manage the queues and compute servers to be used for the remote solution. Any Solve Managers that you have currently defined for RSM will appear in the drop-down list. You can choose one of those Solve Managers or you can specify the name of another machine.
- Queue -- You can select from the queues that you have already defined for RSM.
- **Download Progress Information** -- Specifies that the solver monitor should periodically query RSM for output files in order to display progress (where applicable). Queries to RSM begin when you initiate an action to display the solution progress (such as choosing the Display Monitors option in CFX). This option, available for Fluent and CFX systems, is enabled by default.
 - CFX systems have the following options:
 - → **Always Download Progress Information** causes queries to RSM to begin immediately. This option is enabled by default.
 - → **Download Progress Information on Demand** causes queries to RSM to begin when you initiate an action to display the solution progress (such as choosing the **Display Monitors** option in CFX).
 - Polyflow systems have the following check box:

Download Progress Information – specifies that the solver monitor should periodically query RSM for output files in order to display progress (where applicable). Queries to RSM begin when you initiate an action to display the solution progress. This option is enabled by default.

- **Progress Download Interval** -- Specifies the periodic time interval with which the solver should query RSM for output files in order to display progress. Default is 30 seconds. Setting this value to zero (0) results in continuous queries; that is, as soon as files are downloaded from the compute server, ANSYS Workbench will immediately query again. This option is available for Fluent and CFX systems.
- Execution Mode -- Specifies serial or parallel solver execution mode. The parallel option is available only if the selected solver supports parallel execution mode. This option may not be available with all systems. When performing a design point update via RSM with component update in the foreground, the Parameter Set properties will override the parallel-process settings in individual components. For details on updating design points, see Updating Design Points via Remote Solve Manager(RSM).

If parallel execution is selected for CFX and Fluent, once the job is submitted to RSM:

- If the resulting Compute Server does represents the head node for a cluster (i.e. a third-party scheduler such as LSF, PBS, or HPC is being used), the job is submitted to the scheduler and is executed in distributed parallel across the compute nodes determined by the scheduler, with the specified number of processes being used.
- If the resulting Compute Server does not represent the head node for a cluster, RSM acts as the queuing system and the job is executed in shared memory parallel on the Compute Server, with the specified number of cores being used.
- **Number of Processes** -- If parallel execution mode is selected, specifies the number of processes to use for the solution. Must specify a number greater than or equal to 2.

If a project is unsaved and any cells in your project are configured to use RSM or run in the background, you must save the project or change the solution process settings of those cells to run in the foreground before you can update.

Note

The naming scheme for jobs sent to RSM is as follows:

For Mechanical jobs:

[client machine name]/[Workbench project name]/[design point name]/[Model node name (including cell ID)]/[Environment node name (including cell ID)]/[Solution node name (including cell ID)]

Example: MyWorkstation/MyProject/DP0/Model (A4)/Static Structural (B5)/Solution (B6)

For CFX/Fluent/Polyflow/Mechanical APDL jobs:

[client machine name]/[Workbench project name]/[design point name]/[System name]/[Component name (including cell ID)]

Example: MyWorkstation/MyProject/DP0/CFX/Solution (C3)

Note

System Coupling is not supported under RSM.

The following sections describe how to use the **Solution Process** settings to submit a job to RSM for each of the solvers (where applicable).

Submitting Mechanical Jobs to RSM Submitting Fluids Jobs to RSM Exiting a Project during an RSM Solution Cell Update

Submitting Mechanical Jobs to RSM

See the following sections for more information:

Submitting Mechanical Application Jobs to RSM

Submitting Mechanical APDL Jobs to RSM

Submitting Mechanical APDL Jobs to HPC via RSM

Submitting Mechanical APDL Jobs to PBS via RSM

Submitting Mechanical APDL Jobs to LSF via RSM

Submitting Mechanical Application Jobs to RSM

- 1. Configure your remote solution settings via Remote Solve Manager. See RSM Installation and Configuration for more information.
- 2. From the **Project Schematic**, double-click the **Solution** cell in your Mechanical system to launch the Mechanical application.
- 3. In the Mechanical application, choose **Tools> Solve Process Settings** from the **Main Menu**. Follow the instructions in Using Solve Process Settings to configure your solution settings. The settings established here will be visible in the **Solution Process** properties in ANSYS Workbench.
- 4. In ANSYS Workbench, select the **Solution Process** settings that you want to use for this solution and update the **Solution** cell (via a cell, system, project, or design point update).

Alternatively, you can choose the desired solution configuration from the Mechanical application by selecting **Solve** from the main menu and choosing from the configurations listed in the dropdown list.

Note

- You can Interrupt or Abort an update by right-clicking the **Solution** cell. These options are available during foreground, background, and RSM updates.
- The Workbench Options enable you to control the Mechanical application's license handling. As described in Mechanical (p. 25), you can configure the license to always be released during batch run or to be released when you perform an Update All Design Points run. A third option is On Demand, which makes a Release License option available

on the **Solution** cell's context menu when the cell is in the pending state during a batch run.

Workbench RSM Integration with the Mechanical Application

The properties associated with a Mechanical system's **Solution** cell include a **Solution Process** section. This section enables you to specify the **Solve Process Setting** for the Mechanical application to use when an Update operation is initiated from Workbench. You can select only an existing Solve Process Setting, but new Solve Process Settings can be added (and existing Solve Process Settings can be modified) by selecting **Tools** > **Solve Process Settings** in the Mechanical application.

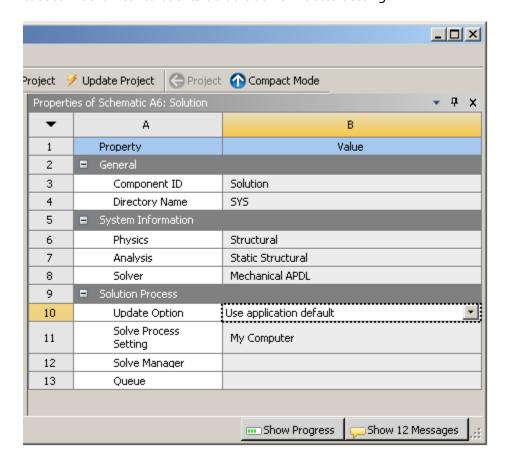
Note

Any solve initiated in the Mechanical application will ignore the Solution component properties selected in Workbench, and will continue to work as they have in previous versions.

Example 1: Default Update with "My Computer" Set as the Default Solve Process Setting

In this case, the default Solve Process Setting in the Mechanical application is "My Computer".

In Workbench, the **Solution Process** property **Update Option** is set to **Use application default**, which causes Mechanical to use its default Solve Process Setting.

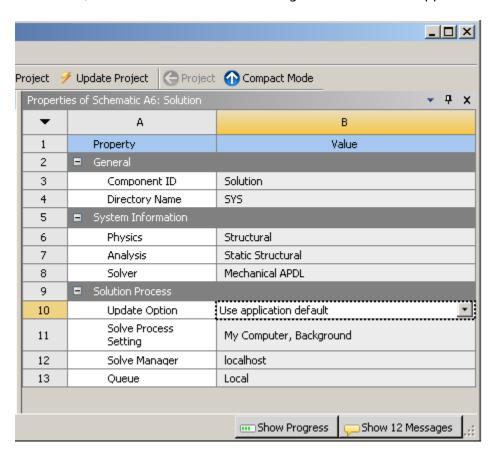


Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer
Mechaical	My Computer

Example 2: Default Update with "My Computer, Background" Set as the Default Solve Process Setting

In this case, the default Solve Process Setting in the Mechanical application is "My Computer, Background".



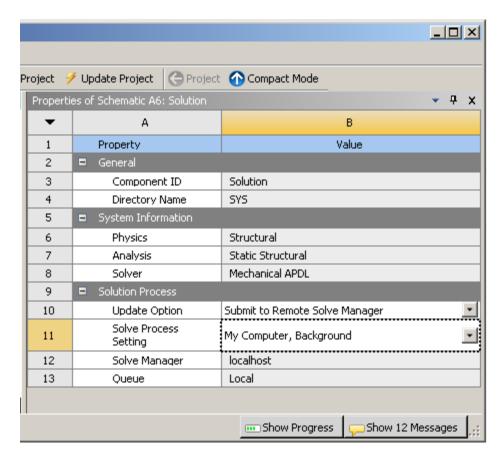
In Workbench, the **Solution Process** property **Update Option** is set to **Use application default**, which causes Mechanical to use its default Solve Process Setting.

Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer, Background
Mechanical	My Computer, Background

Example 3: RSM Update Using "My Computer, Background", with "My Computer" Set as the Default Solve Process Setting

In this case, the default Solve Process Setting in the Mechanical application is "My Computer".



In Workbench, the **Solution Process** property **Update Option** is set to **Submit to Remote Solve Manager**. The Solve Process Setting selected in Workbench is **My Computer**, **Background**.

Results:

Solve Initiated From	Solve Process Setting Used
Workbench	My Computer, Background
Mechanical	My Computer

Submitting Mechanical APDL Jobs to RSM

You can use RSM as a serial or parallel solution for Mechanical PDL jobs. In the **Solution Processes** pane:

- Configure your remote solution settings via Remote Solve Manager. See RSM Installation and Configuration for more information.
- 2. In ANSYS Workbench:
 - a. Set the Update Option to Submit to Remote Solve Manager.
 - b. Set the **Queue**.
 - c. Set the Execution Mode to Parallel.
 - d. Set the **Number of Processes**. For a serial solution, choose 1; for a parallel solution, specify the number of partitions desired ¹.

3. Update the project.

When the Mechanical APDL job is submitted to RSM, some options in the **Properties** view (**Processors**, **Distributed**, **MPI Type**, **Machine List**, and **Custom Executable Path**) become read-only and their values are ignored.

Note

All files involved in a distributed solve will be downloaded, unless **Download Distributed Files** is unchecked.

Note

Mechanical APDL has the following limitations when used with Remote Solve Manager:

- Reconnect will not work after moving a project to another machine.
- Only one copy of a saved project that is in the pending state can reconnect successfully.
- You must manually save a project after a reconnect.
- Reference files must be in the same directory as the ANSYS input file.
- Restarts of solves to LSF, PBS, or HPC are not supported from Workbench.

Submitting Mechanical APDL Jobs to HPC via RSM

If you have a Microsoft HPC queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the **Update Option** to **Submit to Remote Solve Manager**.
- 2. Set the **Queue** to the HPC cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired¹. HPC automatically controls where the partitions are solved.

For details on configuring HPC clusters, see Appendix E in the Remote Solve Manager documentation.

Note

PSSH is not supported on Microsoft Windows for this operation.

Submitting Mechanical APDL Jobs to PBS via RSM

If you have a PBS queue available, you can use that as a parallel solution option. In the Solution Processes pane:

Set the Update Option to Submit to Remote Solve Manager.

- 2. Set the **Queue** to the PBS cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired¹. PBS automatically controls where the partitions are solved.

For details on configuring PBS clusters, see Appendix C in the Remote Solve Manager documentation.

Note

PBS clusters on Windows are not supported.

Submitting Mechanical APDL Jobs to LSF via RSM

If you have an LSF queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the **Update Option** to **Submit to Remote Solve Manager**.
- 2. Set the **Queue** to the LSF cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired¹. The LSF cluster automatically controls where the partitions are solved.

For details on configuring LSF clusters, see Appendix C or Appendix D in the Remote Solve Manager documentation.

Submitting Fluids Jobs to RSM

See the following sections for more information:

Submitting CFX Jobs to RSM

Submitting Polyflow Jobs to RSM

Submitting Fluent Jobs to RSM

Submitting Fluent Jobs to HPC via RSM

Submitting Fluent Jobs to PBS via RSM

Submitting Fluent Jobs to LSF via RSM

Submitting CFX Jobs to RSM

- 1. Configure your remote solution settings via Remote Solve Manager. See RSM Installation and Configuration for more information.
- 2. In ANSYS Workbench, right-click the **Solution** cell and select **Properties**. Select the Solution Process settings that you want to use for this solution and update the project.

Note

If you set Execution Mode to Parallel, you can specify the number of processes that you
want to use to create the results file. You should specify a number that is less than or

- equal to the number of cores available on the Compute Server, and you should ensure that each process contains at least 100,000 nodes or elements.
- You can Interrupt or Abort an update by right-clicking the **Solution** cell. These options are available during foreground, background, and RSM updates.

ANSYS CFX has the following limitations when used with Remote Solve Manager:

- RIF (Flamelet) cases are not supported.
- Reconnect will not work after moving a project to another machine.
- Only one copy of a saved project that is in the pending state can reconnect successfully.
- You must manually save a project after a reconnect.
- You cannot edit a run that is in progress.
- · You cannot perform a remote backup.
- For runs that are submitted to a remote machine:
 - Serial and Local Parallel runs are always supported.
 - Other local parallel modes must be supported on the job host.
 - Distributed parallel is supported only for Compute Server clusters.
- The default update interval for Display Monitors is 120 seconds. Use the **Download Progress Information** and **Progress Download Interval** options under **Tools** > **Options** > **Solution Process** to enable/disable polling or to change the polling interval. The settings established here will also be visible in the Solution Process properties pane in ANSYS Workbench.

Note

This monitoring of solution updates submitted to RSM has the following limitations:

- When you use the **Solution** cell's properties field to set the frequency at which you poll the
 data, this represents a maximum frequency. In cases of network congestion or where large
 files are involved, the observed monitor update frequency will be less.
- Solution monitor data is transferred via the CFX Solver-Manager, which has a limited capacity
 to handle monitor data transfer requests. In situations where multiple users are using the
 same CFX Solver-Manager to monitor runs, or where multiple runs are being monitored by
 a single user, the update frequency of monitor data may decrease or become sporadic.
- When monitoring RSM updates, CFX Solver-Manager will not report run completion. The Out File window will show the completed run information; however, CFX Solver-Manager's workspace will still display "Running". Note that in these cases, Workbench will accurately report update completion.

Submitting CFX Jobs with Design Points to RSM

When performing a Design Point update with a CFX system via RSM, the **Component Execution Mode** and **Max. Number of Processes Per Job** settings on the **Parameter Set** Properties view control the parallel-processing settings that the CFX Solver uses. For details, see Updating Design Points via Remote Solve Manager (RSM).

Submitting CFX Jobs to HPC via RSM

If you have a Microsoft HPC queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the Update Option to Submit to Remote Solve Manager.
- 2. Set the **Queue** to the HPC cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired ¹. The HPC cluster automatically controls where the partitions are solved.

For details on configuring HPC clusters, see Appendix E in the Remote Solve Manager documentation.

Submitting CFX Jobs to PBS via RSM

If you have a PBS queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the **Update Option** to **Submit to Remote Solve Manager**.
- 2. Set the Queue to the PBS cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired¹. PBS automatically controls where the partitions are solved.

For details on configuring PBS clusters, see Appendix C in the Remote Solve Manager documentation.

Note

PBS clusters on Windows are not supported.

Submitting CFX Jobs to LSF via RSM

If you have an LSF queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the **Update Option** to **Submit to Remote Solve Manager**.
- 2. Set the **Queue** to the LSF cluster.
- 3. Set the **Execution Mode** to **Parallel**.

4. Set the **Number of Processes** to the number of partitions desired¹. The LSF cluster automatically controls where the partitions are solved.

For details on configuring LSF clusters, see Appendix C or Appendix D in the Remote Solve Manager documentation.

Submitting Polyflow Jobs to RSM

- 1. Configure your remote solution settings via Remote Solve Manager. See RSM Installation and Configuration for more information.
- 2. In ANSYS Workbench, right-click the **Solution** cell and select **Properties**. Select the Solution Process settings that you want to use for this solution and update the project.

Note

- You can Abort an update by right-clicking the **Solution** cell. This option is available during foreground, background, and RSM updates.
- Polyflow has the following limitations when used with Remote Solve Manager:
 - Reconnect will not work after moving a project to another machine.
 - Only one copy of a saved project that is in the pending state can reconnect successfully.
 - You must manually save a project after a reconnect.
 - The RSM Interrupt option performs an abort operation, not an interrupt.
 - Parallel design point submissions to batch queue clusters will only run on the master node and with the number of cores allocated by the batch queue scheduler.
 - Polyflow jobs submitted via RSM are not supported on SGE clusters.

Submitting Fluent Jobs to RSM

You can use this feature to queue multiple jobs to run on the local machine, such as overnight or during other low-usage times, and you can submit a job to remote machines.

- 1. Configure your remote solution settings via Remote Solve Manager. See RSM Installation and Configuration for more information.
- 2. In ANSYS Workbench, right-click the **Solution** cell and select **Properties**. Select the Solution Process settings that you want to use for this solution and update the project.

¹You should have at least 100,000 elements or nodes per partition to compensate for the overhead associated with the partition.

An update of the **Solution** cell submits the job to RSM, moves into Pending mode for the duration of the solution, and then automatically reconnects at the end of the run.

Tip

If you set **Execution Mode** to **Parallel**, you can specify the number of processes that you want to use to create the results file. You should specify a number that is less than or equal to the number of cores available on the Compute Server, and you should ensure that each process contains at least 100,000 nodes or elements.

Note

- You can Interrupt or Abort an update by right-clicking the **Solution** cell. These options are available during background and RSM updates.
- Fluent has the following limitations when used with Remote Solve Manager:
 - Only one copy of a saved project that is in the pending state can reconnect successfully.
 - System Coupling is not supported.
 - UDFs are supported but you must have a supported compiler on Windows 32-bit and 64-bit machines. Supported compilers for Windows are Microsoft Visual Studio 2008 Standard and Microsoft Visual Studio 2010 Professional.
 - On Linux, UDFs are supported. You can always send UDFs between Linux machines and make use of the auto-compile feature; if the machines have compatible compilers, you can send precompiled UDFs.

For additional information about compiling Fluent UDFs, see Compiling UDFs in the Fluent UDF Manual.

Solution Properties: Use Setup Launcher Settings

The **Properties** view of the Fluent **Solution** cell has a toggle that controls whether the solution uses settings from the Setup Launcher. When **Use Setup Launcher Settings** is checked, the Launcher Settings are copied from **Setup** component to **Solution** component. This copying also happens any time you change the Launcher settings in setup. When **Use Setup Launcher Settings** is cleared, the **Solution** cell's Launcher settings are available:

General

Precision

Default: Single Precision

Applicable to RSM: Yes

Show Launcher at Startup

Default: Enabled

Applicable to RSM: No

Display Mesh After Reading

Default: Enabled

Applicable to RSM: Yes

Embed Graphics Window

Default: Enabled

Applicable to RSM: Yes

Use Workbench Color Scheme

Default: Enabled

Applicable to RSM: Yes

Setup Compilation Environment for UDF

Default: Enabled

Applicable to RSM: No

Use Job Scheduler

Default: Disabled

Applicable to RSM: No

Run Parallel Version

Default: The initial default execution mode (Serial / Parallel) is based on the Run Settings. The initial 'Number of Processes' in RSM is based on the **Number of Processors** option in the run settings: if it is > 1, that value will be used, otherwise 2 will be used. If **Run Parallel Version** is not enabled, then Serial mode is forced.

Applicable to RSM: Yes

UDF Compilation Script Path

Default: \$(FLUENT_ROOT)\\$(ARCH)\udf.bat

Applicable to RSM: No

Use Remote Linux Nodes

Default: Enabled

Applicable to RSM: No

Remote

Remote Polyflow Root Path

Default: none

Applicable to RSM: No

Use Specified Remote Working Directory

Default: Enabled

Applicable to RSM: No

Remote Working Directory

Default: none

Applicable to RSM: No

Remote Spawn Command

Default: RSH

Applicable to RSM: No

Use Remote Cluster Head Node

Default: Enabled

Applicable to RSM: No

Remote Host Name

Default: none

Applicable to RSM: No

Parallel Run Settings

Number of Processors

Default: 1

Applicable to RSM: No

Interconnect

default

Applicable to RSM: No*

Note

* You can enable **Interconnect** to be available for Polyflow RSM runs by enabling **Tools** > **Options** > **Solution Process** > **Show Advanced Solver Options**. This setting requires you to ensure that the remote Compute Servers can accept the Interconnect that you specify—there is no automatic checking for such compatibility.

To learn what values are available, see Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options in the *Fluent User's Guide*.

MPI Type

default

Applicable to RSM: No*

Note

* You can enable **MPI Type** to be available for Polyflow RSM runs by enabling **Tools** > **Options** > **Solution Process** > **Show Advanced Solver Options**. This setting requires you to ensure that the remote Compute Servers can accept the MPI Type that you specify—there is no automatic checking for such compatibility.

To learn what values are available, see Starting Parallel ANSYS Fluent on a Windows System Using Command Line Options in the *Fluent User's Guide*.

Use Shared Memory

Default: Disabled

Applicable to RSM: No

Machine Specification

Default: (file containing the machine list)

Applicable to RSM: No

Machine Filename

Default: none

Applicable to RSM: No

The **Solution** cell's **Solution Process** setting is always displayed; when **Use Setup Launcher Settings** is checked, the option is read-only:

Solution Process

Update Option

You can choose to **Run in Foreground**, **Run in Background**, or **Submit to Remote Solve Manager**. When **Submit to Remote Solve Manager** is chosen, options that are not applicable to RSM are hidden. Also, fluentlauncher.txt will not contain options that conflict with RSM.

Submitting Fluent Jobs to HPC via RSM

If you have a Microsoft HPC cluster queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the Update Option to Submit to Remote Solve Manager.
- 2. Set the **Queue** to the HPC cluster.
- 3. Set the **Execution Mode** to **Parallel**.
- 4. Set the **Number of Processes** to the number of partitions desired ¹. The HPC cluster automatically controls where the partitions are solved.

For details on configuring HPC clusters, see Appendix E in the Remote Solve Manager documentation.

Submitting Fluent Jobs to PBS via RSM

If you have a PBS queue available, you can use that as a parallel solution option. In the Solution Processes pane:

- 1. Set the **Update Option** to **Submit to Remote Solve Manager**.
- 2. Set the **Queue** to the PBS cluster.
- 3. Set the Execution Mode to Parallel.

4. Set the **Number of Processes** to the number of partitions desired ¹. PBS automatically controls where the partitions are solved.

For details on configuring PBS clusters, see Appendix C in the Remote Solve Manager documentation.

Note

PBS clusters on Windows are not supported.

Submitting Fluent Jobs to LSF via RSM

If you have an LSF queue available, you can use that as a parallel solution option on Linux. In the Solution Processes pane:

- 1. Set the Update Option to Submit to Remote Solve Manager.
- 2. Set the **Queue** to the LSF cluster.
- 3. Set the Execution Mode to Parallel.
- 4. Set the **Number of Processes** to the number of partitions desired ¹. The LSF cluster automatically controls where the partitions are solved.

For details on configuring LSF clusters, see Appendix C or Appendix D in the Remote Solve Manager documentation.

Exiting a Project during an RSM Solution Cell Update

You can exit a project while a **Solution** cell update via RSM is in progress (i.e., one or more RSM jobs are running in the background and the **Progress** view displays a project **Status** of **Waiting for background task**).

In order for RSM jobs to continue to run after you exit the project, the project must be saved at least once after the **Solution** cell update job was initiated.

If you attempt to exit a project while a **Solution** cell update job is still running, the following scenarios will cause a dialog to display, allowing you to specify whether you want to save the project before exiting:

- You have never saved the project at any time after the same Solution cell update job was initiated.
- You have saved the project at least once after the same **Solution** cell update was initiated, but results have been retrieved since your last Save.

In either of these cases, if you do not save the project before exiting:

- All results retrieved since the last Save operation will be lost. You must save the project before exiting if you want to keep any of results retrieved since the last save of the background **Solution** cell update.
- If the project has never been saved after the **Solution** cell update job was initiated, all RSM jobs will be aborted and will show a **Status** of **Cancelled** and the Cancelled icon () in the RSM **List** view; the asterisk on the icon indicates that the job has also been released. When you reopen the project, it will be in the state of your last manual save.

• If the project has been saved at least once after the **Solution** cell update job was initiated, RSM jobs that are queued and running will continue to run after you exit. When you reopen the project, you can resume the update of the pending jobs to reconnect and download the results.

If you do save the project before exiting:

- Retrieved results will be saved to the project.
- RSM jobs that are queued and running will continue to run after you exit.
- Jobs for which the results have been saved will show a **Status** of **Finished** and the Finished icon in the RSM **List** view; the asterisk on the icon indicates that the job has also been released.
- Jobs for which results have not yet been saved are not released upon exit. When you reopen the project, you can resume update of the pending jobs to reconnect and download the results.

Using Journals and Scripts

ANSYS Workbench offers the ability to record the actions you perform via the GUI, which we refer to as *journaling*. Journals are recorded as Python-based scripts. You can modify these scripts or create new ones, which we refer to as *scripting*. Together, these capabilities allow you to quickly and easily replay analyses you've already run via recorded journals, as well as to extend functionality, automate repetitive analyses, and run analyses in batch mode.

For detailed instructions on using journaling and scripting, as well as a complete list of all available data containers, namespaced commands, and data types, see the *Workbench Scripting Guide*.

Related Topics:

Journaling Scripting

Journaling

A journal is a record of all operations that have modified data during your session. Based on your Preferences setting, a journal of your full session will automatically be saved to the location you specify (see Setting Journaling Preferences). You can also choose to record part of a session to a journal file, capturing a specific set of actions. Playing back the journal will recreate the recorded actions exactly. Journaling and scripting tools (including recording and playback) are available through the **File**> **Scripting** menu and can be performed by anyone using the ANSYS Workbench UI.

- Use journaled sessions to restore work after a crash.
- Journals are platform independent and portable, subject to file location consistency between accounts (see File Path Handling in ANSYS Workbench for details on file path handling within journals and scripts). They can be used with any ANSYS installation (release 12.1 or higher).

Setting Journaling Preferences

You can set journaling preferences such as the default directory where journals will be written and how long to keep a journal file.

- 1. In ANSYS Workbench, select **Tools> Options> Journals and Logs**.
- 2. Select **Record Journal Files** to have ANSYS Workbench automatically write journal files.

- 3. Specify the default location where journal files will be written. This is the location that the browser will open in automatically when you choose to begin recording a journal. You will still be able to browse to a different location before saving a particular journal.
- 4. Specify the number of days to keep a journal file.
- 5. Specify how long (in seconds) to pause between each command when running a journal file.
- 6. Click **OK** to save your settings.

Recording and Playing a Journal

Follow the steps described below to record a journal and then to playback a journal interactively. To use the command window, see Using the Command Window (p. 71).

Recording a Journal

- 1. Launch ANSYS Workbench.
- 2. Select File> Scripting> Record Journal.
- 3. Specify the name and location of the journal file and click **Save**.
- 4. Use the GUI to work through your analysis as you normally would.
- 5. Select File> Scripting>Stop Recording Journal.
- 6. A message appears informing you that you will stop recording. Click **OK**.

Note

Not all actions are journaled. Only actions that change project data are journaled. Some examples of actions that are not journaled include:

· GUI-only actions, such as:

interrupting a Solve operation running in Compact mode launching help (including quick help and the **Sidebar Help** view) running the View Solver Output option from VistaTF's **Solution** cell

 Actions taken in some data-integrated applications; see Scripting and Data-Integrated Applications.

Playing Back a Recorded Journal

- 1. Select File> Scripting> Run Script File.
- 2. Select the journal file to be played back and click **Open**.
- 3. The previously recorded actions will occur.

Using the Command Window

The command window allows you to invoke commands, access data entity properties, and invoke data entity and data container methods interactively, one at a time.

- Select File> Scripting> Open Command Window.
- 2. Enter the commands you want to run, one at a time.
- 3. As you enter each command, the appropriate action will occur in the ANSYS Workbench GUI.

The Console Window The console window is the same as the command window but is present when running in batch mode to provide a way of working directly with commands outside of the user interface.

Scripting

A script is a set of instructions to be issued to ANSYS Workbench. The script can be a modified journal, or it can be a completely new set of instructions that you write directly.

The creation of scripts requires a general understanding of programming constructs and paradigms. ANSYS Workbench uses an object-based approach, similar to object-oriented programming.

For detailed information on using Scripting, see Using Scripting in ANSYS Workbench in the *Workbench Scripting Guide*.

Project File Management

ANSYS Workbench's file management system stores several different files under a single project, using directory trees to organize files relevant to each system and the applications used in the system.

When the project file (<filename>.wbpj) is created, ANSYS Workbench creates a project folder named <filename>_files where <filename> is a name you provide. All files relevant to the project are saved within this folder.

The primary subdirectories within the project folder are dp0, dpall, and user files.

We strongly recommend that you use caution when directly modifying any of the content in any of the ANSYS Workbench project directories or subdirectories other than user_files. You should work through the ANSYS Workbench GUI to manage your project as much as possible. ANSYS Workbench may not recognize or be aware of any changes that you make directly in the file system (such as adding or removing a file).

Project Directories

The project directory structure includes the follow directories:

dp0 Subdirectory (p. 72) user_files Subdirectory (p. 72) dpall Subdirectory

dp0 Subdirectory

ANSYS Workbench designates the active project as design point 0 and creates a dp0 subdirectory that always corresponds to the active project files. For more information on dp0 and design points, see the section dpall Subdirectory and Working with Design Points (p. 73).

Within the design point folder are system folders for each system in the project. Within each system folder are folders for each application used in the system (i.e., the Mechanical application, Fluent, etc.). These folders contain application-specific files and folders, such as input files, model directories, engineering data, resources, etc. System folders for each system type are named as follows.

System Type	Folder name
Autodyn	ATD
BladeGen	BG
Design Exploration	DX
Engineering Data	ENGD
FE Modeler	FEM
Fluid Flow (Poly-	FFF (analysis system), FLU (component sys-
flow)	tem)
Fluid Flow (CFX)	CFX
Geometry	Geom
Mesh	SYS (top level) / MECH (subdirectory)
Mechanical	SYS (top level) / MECH (subdirectory)
Mechanical APDL	APDL
TurboGrid	TS
Vista TF	VTF
Icepak	IPK

^{*} The Mechanical application and Mesh system folders under the dp folder(s) are labeled SYS. Both the Mechanical application and Mesh files are written to MECH subdirectories, because both are generated by Mechanical-based applications

In addition to the system folders, the dpn folders also contain a global folder. This folder contains subdirectories for all systems in the project. These subdirectories may be shared by more than one system in the project and contain all database files, as well as any files that are associated directly with the database files. For example, the Mechanical application will write figures and images and contact tool data to the appropriate system subdirectory under the global folder.

user_files Subdirectory

Also under the project folder is a user_files directory. This folder contains any files (such as input files, referenced files, etc.) that you supply to a project or any output (images, charts, movie clips, etc.) generated by ANSYS Workbench that you wish to have associated with the project. In most cases, you are responsible for placing required files into this directory. In other cases, such as the export of design point update data from a design exploration system to a CSV log file, data is written directly to a file created in this directory. For more information on design point data being exported to a CSV file, see Extended CSV File Format in the DesignXplorer help.

Along with other project files, all of the files contained in the user_files directory appear in the **Files** view in ANSYS Workbench. Thus, any files that you have placed into this directory can be accessed easily from the ANSYS Workbench user interface via the **Open Containing Folder** option of the right-click context menu.

ANSYS Workbench also protects this directory and ensures that it is managed and archived appropriately with the rest of the project; therefore, you can safely store additional files (such as PowerPoint or Excel files, or other files from separate applications that are associated with this project) here without the risk of losing data. If you save files in any other directory in the project structure and then exit without saving, ANSYS Workbench will delete any files saved there since the last ANSYS Workbench save.

dpall Subdirectory and Working with Design Points

ANSYS Workbench allows you to create multiple design points and generate comparison studies of input and output parameters. To analyze your simulation across several design points, you must first generate input parameters for the current project (and, if appropriate, specify output parameters to be created). Once you have created parameters, you will see a **Parameters** cell added to the relevant system(s) and a **Parameter Set** bar added to the project. At this point, the current project is designated as Design Point 0, or dp0.

Use the Parameters tab to vary input parameters and create multiple design points, which can be updated separately or sequentially. Before running a design point update, you should decide whether you want to retain the files generated during the design point update for further analysis. ANSYS Workbench saves only the data for the current design point and the values of the output parameters computed for each design point. If you want a different design point to be the current design point, you can use the **Copy inputs to Current** option on that design point's context menu. If you want to save the data files for another design point(s), you must mark it for export before updating that design point(s).

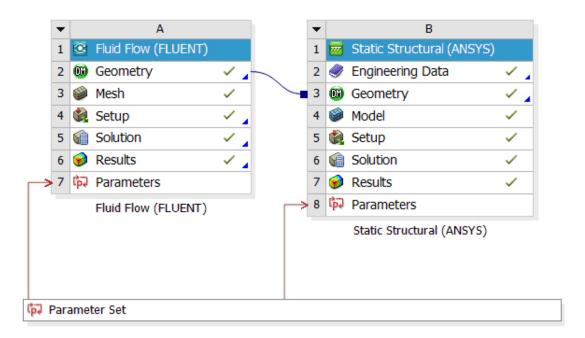
While the design point update is running, ANSYS Workbench creates temporary design point folders. If you select a design point for export, the temporary design point folder is saved as a new project, named <filename>_dpn_files, where dpn indicates the design point number you exported. This folder will be a sibling to the original <filename>_files project folder. To export a design point, you must mark it for export before updating it; if a design point is already up-to-date and is then marked for export, you must update the design point again in order to export it. For projects that include a Design Exploration system, ANSYS Workbench creates a dpall folder containing the input and output parameters when a design point update occurs.

If you change a parameter in an exported design point and then run an update, the exported project files will be overwritten. However, if you first deselect the **Exported** check box, then change the design point, and then run an update, the exported project will not be overwritten but will no longer be consistent with the data in the **Table of Parameters**.

For more details on working with parameters and design points, see Working with Parameters and Design Points (p. 93).

Example Project

A finished project that includes a Fluid Flow (Polyflow) system (FFF), a Mechanical application system (MECH), and parameters (DesignXplorer) might look like this:



The corresponding directory structure would look like this:

```
Myworkbenchproject_files
 dp0
   FFF
     DM
     Fluent
     MECH
     Post
   qlobal
     MECH
       FFF
       SYS
   SYS
     ENGD
     MECH
 dpall
   qlobal
```

Working with Files and Projects

See the following sections for more information on:

Importing Files Archiving Projects Project Recovery Project Locking

Importing Files

When working in ANSYS Workbench, you may need to import files, such as input files, existing mesh files, geometries, etc.

When you edit an imported file, ANSYS Workbench saves a copy of the file to the project directory, rather than overwriting the original file. This process ensures that your original files are never compromised.

Most files from previous releases can be imported using **File> Import**. The import operation will create systems, cells, and links to represent the previous release project. For more information on importing legacy databases, see Importing Legacy Databases (p. 78).

Archiving Projects

If you want to send a project to a colleague or to ANSYS Technical Support, or need to package all of the files for archiving or other purposes, choose **File> Archive**. In the **Save Archive** dialog box, navigate to the directory where you want to save the file and select the archive type: Workbench Project Archive (.wbpz) or a Zip (.zip/.tar.gz) file. You will also need to specify which optional items you want to archive, such as result/solution items, imported files, and items in the user files directory.

The Workbench **Options** dialog allows you to specify the compression level for .wbpz archives. For more information, see Project Management (p. 14).

When you import an external file, Workbench archives it by adding it to the user_files directory. However, if the external file refers to other files (for example, as when a CAD assembly is linked to the CAD parts), the system is not able to place all of the necessary referenced files in the user_files directory. In order to make the referenced files part of the archive, you must copy them manually into theuser_files directory.

To restore an archived file, select **File> Restore Archive**. You will be prompted for a project path to which the archive will be extracted, and then that project will be opened.

Note

If your project contains Imported Boundary Conditions in the Mechanical application, you should choose to include result/solution items so that the necessary upstream files are archived. Failure to archive these files will prevent you from importing data or accessing features that involve reading upstream data, when the project is restored.

Windows Only On Windows systems, you can also double-click the .wbpz file to open the archive. If you double-click the .wbpz to open a file and then make changes to the project, when you save the project, you will be prompted to either overwrite the archive, create a copy of the archive, or cancel the save operation. If you choose to create a copy of the archive, you will be prompted for a name and location for the copy. The new copy will also have a .wbpz extension. After the save operation, you will be returned to ANSYS Workbench, working in the new (copy) archive. The original archive will remain unchanged. Use **File> Save As** to restore the project to a .wbpj file.

You cannot update retained design points when working in an archived project. If you choose to update a design point with the retained option on, you will be prompted to first use **File> Save As** to save the project as a .wbpj file.

Project Recovery

ANSYS Workbench creates backup files of projects that are currently active and in progress. In the event of a crash, ANSYS Workbench can use these backup files to restore your project to the last saved event. As with any computer program, it's important that you save your work frequently to minimize data loss in the event of a crash. Do not move or otherwise alter the backup directory.

If a project save operation fails (for example, an application is busy and cannot execute the save), you will be given the following options:

- · Revert to the last saved project.
- Make a copy of the last saved project before continuing with the partially saved project. The project will be copied into a new location that you specify.
- Exit Workbench and decide later. Use this option if you wish to handle the save failure manually. As a
 result, the backup directory will NOT be cleared so that you can manually recover files from that directory
 later.
- Continue with the partially saved project, discarding the last saved project (not recommended). This option
 results in the backup directory being cleared. Use this option with caution, as it could result in corrupt
 project files.

Note

This save-failure behavior applies only to a Save operation and NOT to a Save As (or first save) operation.

Project Locking

ANSYS Workbench implements a project locking mechanism in order to help prevent the project from being loaded into more than one session at a time. A project is locked by creating a .lock file in the project files directory. The project is unlocked by deleting the .lock file.

An improperly unlocked project can occur in situations such as program crash where the .lock file is not deleted, or if the project files directory is duplicated and the .lock file is copied with it. If ANSYS Workbench finds a project is locked, you will be asked how to proceed.

If the file is locked because of an abnormal termination, such as a program crash, you can safely select Unlock and continue. If the file is locked because the project is already open in another ANSYS Workbench session, you should select Cancel. Opening the same project in multiple sessions can result in corrupted project files.

Notes About Project File Management

We strongly recommend that you use caution when directly modifying any of the content in any of the ANSYS Workbench project directories or subdirectories other than user_files. You should work through the ANSYS Workbench GUI to manage your project as much as possible. ANSYS Workbench may not recognize or be aware of any changes that you make directly in the file system (such as adding or removing a file).

If you have deleted any project files through the file system and not through the ANSYS Workbench GUI, you may see a warning message when you attempt to open the project. The error message will list the missing file(s). The corrective action depends on the type of files that are missing:

For files that are programmatically integrated with ANSYS Workbench (such as database files, geometry files, engineering data files, application-generated files, etc.), use View > Files to open the Files pane, where you can use the context menu to repair a file or permanently remove it from the project's file list if it is no longer needed. Repairing these types of files should be done judiciously; it is possible to use files or file types that are not similar to the original. Missing or erroneous files can cause unintended consequences for project stability and usability.

• For files that are not programmatically integrated with ANSYS Workbench (such as user-supplied input files, dpn files, etc.), you can replace them using your file system and then open the project again. These files may not be necessary for the project; in this situation, you can safely ignore the warning message.

The project system may delete, back up, or restore files when:

- deleting, duplicating, or replacing systems in the schematic
- opening an application for a cell
- · closing a project without saving
- archiving a project
- switching to the next design point during the execution of the Update All Design Points operation

While the project system is performing one of these operations, you cannot have project files open in other applications (such as a text editor) or have the directories open in Windows Explorer. Doing so may cause these file management operations to fail.

You cannot move a project or any of its associated files to another machine while a background run is in progress. File information for the background run is, by necessity, machine-specific. You cannot package or modify the background run while it is in progress.

You will also have errors if you move a project that has references to files outside of the project directory to a different machine or location. By opening the project from a different machine or location, those file references will no longer resolve unless the file is still available under the same absolute path.

Copied images exist in only one location on disk that is referenced and do not exist as physical copies. If you delete an image that has been copied, all pointers to the copies of that image will contain broken links.

ANSYS Workbench Files

To view all files associated with a project, choose **View>Files** from the menu bar. You will be able to see the name and type of file, the ID of the cell the file is associated with, the size of the file, the location of the file, and other information. Files added to the project will appear here. Files deleted from the project will be shown in red and will be marked with a "Deleted" icon. See Files View (p. 247) for more information on using the **Files** view. Database files associated with the various ANSYS Workbench applications are listed below.

Database Files ANSYS Workbench applications create the following types of database files:

- ANSYS Workbench project database file = .wbpj
- Mechanical APDL = .db
- Fluent = .cas, .dat, .msh
- CFX = .cfx, .def, .res, .mdef and .mres
- DesignModeler = .agdb
- CFX-Mesh = .cmdb

- Mechanical = .mechdb
- Meshing = .cmdb
- Engineering Data = .eddb
- FE Modeler = .fedb
- Mesh Morpher = .rsx
- ANSYS Autodyn = .ad
- DesignXplorer = .dxdb
- BladeGen = .bqd

Workbench Journal and Log Files ANSYS Workbench writes journal and log files for each ANSYS Workbench session. For more information on journal and log files, see Journals and Logs (p. 17).

Design Point Log Files During a design point update, the parameter values of each successfully updated design point are written to a CSV log file in the user_files directory. For more information, see Design Point Log Files in the DesignXplorer help.

Importing Legacy Databases

ANSYS Workbench offers several methods to import databases from earlier releases:

- Context Menu Import: The file can be imported via a cell's context menu. To import a file using this
 method, you first create the appropriate system and cell, and then import the file into the cell via the
 context menu.
- 2. File > Import: The file can be imported via the File > Import menu. The proper systems and cells are created and populated with the data from the imported file. However, you must launch the associated application or editor to use the imported file. You can also drag-and-drop one or multiple files from Windows Explorer onto the Project Schematic. These files will be treated as if they were imported via File > Import.
- 3. **Project Import:** The file can be imported as part of an earlier release's project import. The file must be listed as part of the earlier release's project file's contents. When the project file is selected via the **File** > **Import** menu, all necessary systems, cells, and links will be established and populated with data from the various files that made up the earlier release's project.

In each of these cases, you must launch the associated application to use the imported file, which is typically the same application that was used to edit the file in the earlier release. In other words, the imported files are not undergoing any transformation in this release of ANSYS Workbench. Rather, access to the files is being coordinated through the **Project Schematic** interface.

Not all products/databases use all of these methods.

DesignXplorer Release 11 (.dxdb) file import is not supported; however, basic parameter import is supported.

The following table shows which applications can use the three methods described above.

Table 1: Import Methods

Application	Context Menu Import (#1)	File>Import (#2)	Project Import (#3)
ANSYS Workbench, Release 11 or 10 (.wbdb)			X
Mechanical (.dsdb)		X	X
FE Modeler (.fedb)		Х	X
Engineering Data (.eddb)			Х
			(except convections and load histories)
Engineering Data (.xml, .engd)		X	
AWA (.aqdb)		X	X
Autodyn (.ad)		Х	X
Mechanical APDL (.inp,.dat,.cdb,.mac,.anf)	Х		
DesignModeler (.agdb)	Х	Х	X
Meshing ^a (.cmdb)		Х	X
Fluent (.msh, .cas, .dat)	X		
CFX(.cfx,.def,.res)	X	X	
BladeGen (.bdg)	Х	Х	

a. cmdb files from Release 10 that contain CFX-Mesh data are not supported. To import these files, import them first into Release 11, save as a Release 11 file, and then import into Release 15.0.

Additional Mechanical Application Import Details

- Legacy .dsdb files that contain multiple models are split into multiple files.
- .dsdb files become .mechdb files internally.
- Separate systems are created to correspond to each of the Release 11 environments. Environments that
 use the same geometry and model are represented with a link between the **Geometry** and **Model** cells
 of the systems. Physics environments are typically represented as a link between a **Solution** cell of the
 originating system and the **Setup** cell of the receiving system. For example, a thermal condition is represented by a connection between the **Solution** cell of a thermal system and the **Setup** cell of a structural
 system.
- Information that is transferred from the Release 11 system to the current system includes:
 - model name
 - model state
 - physics type
 - internal IDs for model, mesh, environment, and answer set

- Engineering Data file path
- material property suppression path
- solver files directory path
- solver type
- geometry preferences
- parameters
- Legacy .dsdb files that do not contain an environment are imported as Mechanical Model systems.

Additional FE Modeler Import Details

- FE Modeler systems will be imported as standalone systems. You can manually create any necessary links between systems, but you will not have the data that FE Modeler may have created (such as geometry).
- If FE Modeler was associated with a Release 11.0 applet, and the .dsdb being imported does not contain any environments, a link is established between the **Model** cell of the Mechanical Model system and the **Model** cell of the FE Modeler system. Likewise, if FE Modeler was associated with a Release 11.0 Meshing applet, a link is also established.
- FE Modeler allows you to import a number of mesh files through the **Model** cell context menu.

Additional Aqwa Import Details

- A legacy Agwa database is imported as a standalone Hydrodynamic Diffraction system
- If the legacy database contains multiple analyses, these will be converted into multiple Hydrodynamic Diffraction systems
- Geometry associated with the legacy database will be associated with the Geometry cell of the Hydrodynamic Diffraction system and will be editable

Additional Autodyn Import Details

No links are created when Autodyn files are imported from Release 11.0.

Additional CFX Import Details

If CFX files are present in the .wbdb project, they will appear in the **Files** view, but no system or links associated with these files are created. You can choose to **Import to Schematic** from the **Files** view, which will create a CFX system and import the selected file.

You can import CFX-Solver .bak files or full .trn files into the **Project Schematic** so that you can post-process results for debugging when a run fails. You may find it useful to use full .trn files as a backup mechanism because all timesteps are retained, instead of just the most recent ones, and it is easier to post-process multiple timesteps.

Additional DesignModeler Import Details

• DesignModeler files (.agdb) can be imported through context menu import.

- agdb and .modeldv files can be imported via File> Import.
- Links with other cells are automatically generated when .agdb files that are present in the Release 11.0 .wbdb project are imported into the appropriate system (Mechanical Model or other appropriate system).
- Parameters contained in the .agdb file will not be immediately published to the **Project Schematic** interface. They will be published when the DesignModeler application is opened.
- CAD files that were imported into the .agdb will not immediately be registered in the **Files** view of the **Project Schematic**. Registration of these files will occur when the DesignModeler application is opened.

Additional Meshing Import Details

- Links with other cells are automatically generated when .cmdb files that are present in the Release 11.0 .wbdb project are imported into the appropriate system (Mechanical model or other appropriate system).
- CFX-Mesh files (.gtm, .cfx) can be imported via context menu import from the Mesh cell of a Mesh system.

Working with the Chart View

The **Chart** view is available for Workbench applications. Applications can display data using a number of different basic chart types. Each chart type has editable display properties.

The following chart topics are discussed:

Chart Types
Setting Chart Properties
Chart Zoom, Pan, and Rotate
Using the Triad
Saving a Chart

Chart Types

Although the content of a chart will be tailored to the application that is displaying the chart, there are a standard set of charts that you might see in Workbench. These include:

XY Plot

Lines, points, steps, bars, splines, scatter, or shaded regions can be plotted.

XYZ Plot

Lines, points, bars, lines, splines, scatter, or surfaces can be plotted.

Pie Chart

Displays multiple variables as corresponding segments in rings on the pie.

Spider Chart

Similar type of display as the pie chart, in multiple dimensions. Good for displaying between 3 and 10 variables. Can usually be displayed as a Parallel Coordinate plot also.

Parallel Coordinate Plot

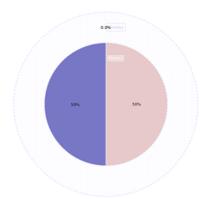
Provides a graph to display variables (design points, etc.) using parallel Y axes to represent all of the inputs and outputs. Selecting an axis allows you to filter the variables shown by dragging the arrows at the ends of the axis (variables with points outside of the axis range will be eliminated from the chart). Can usually be displayed as a Spider chart also.

Correlation Matrix

Allows you visualize how closely the various input and output parameters are coupled. The strength of correlation is indicated by color in the matrix.

Note

If one of the variables to be displayed in a pie chart has no values associated with it, a ring like the outer one in the image below will appear for that variable.



Setting Chart Properties

Each chart has properties that can be set for the chart data, and properties that can be set for the chart display. The chart data properties should be discussed in the help for the individual applications that are displaying the chart. This section will talk about the chart display properties, which should be common to charts displayed by any Workbench application. The various chart properties will be shown in the **Properties** view in the tab.

Note

Only the properties that are applicable for the current chart type will be displayed when you edit any particular chart component (an axis, for example). Therefore, although a property may be listed in this document under Axis Properties, that property might only appear for one or two chart types, or the property may only appear for continuous or discrete axes.

Axis Properties

To set axis properties, right click on or outside of an axis and select **Edit Properties**. The axis properties that are available will depend on whether the chart axis is discrete or continuous. Some of the settings include:

- Title name for that axis (often defaults to variable name)
- Title Background Color background color used for axis name
- Show/Hide Grid if checked, show a grid for this chart axis
- Automatic Range if checked, use automatic scaling for the axis, otherwise use the Range Minimum and Range Maximum, if they fall within the data bounds
- Range Minimum set minimum range of axis

- Range Maximum set maximum range of axis
- Is Logarithmic if checked, set axis scaling to be logarithmic if (linear if unchecked)
- Is Usability if checked, display a line/bar plot of the cross section data in a perpendicular direction to the axis direction

Legend Properties

Legend properties can be set by right clicking on the legend (border or background) and selecting **Edit Properties**, or can be set as part of the general chart properties. Be sure not to click on one of the legend variables, since the properties for that variable will be displayed, rather than the legend properties. The legend properties include:

- Visible if checked, displays the legend for the chart
- Style displays the legend entries either horizontally or vertically, and allows the legend to be expanded in the same direction, if it provides any benefit (defaults to vertical)
- Foreground Color sets the color of the legend border
- Background Color sets the color of the legend background

The legend will default to no background color or foreground color (border). When you hover the cursor over the legend, you should see its borders. You set the background and foreground colors using a color wheel. You can select a new color and apply it, and you can also click **More>>** and change the **Alpha** channel to 0 for transparent or 255 for opaque.

Note

If there are too many entries in the legend, the legend will not display, even if Visible is checked.

Variable Properties

Variable display properties can be set by either right clicking on the variable plot on the chart and selecting **Edit Properties**, or by right clicking on the variable name in the legend and selecting **Edit Properties**. The **Properties** view will show the display properties available for that variable. The properties, when available, may include the following fields.

- Label name for that variable plot
- Display As selects the type of plot used for the variable
- Automatic Range set automatic determination of variable range on or off
- Range Minimum set minimum range of variable values displayed
- Range Maximum set maximum range of variable values displayed
- Allow Filtering allow filtering on parallel coordinate plot

Style Display Properties

When plot rendering is controlled per variable, the **Style** display properties will appear when you edit the variable properties. When plot line rendering is controlled generally rather than per variable (as for spider charts, for example), the **Style** display properties will appear when you edit the chart properties.

- Smoothed Edges renders lines/surfaces with anti-aliasing set on, such that the line/surface appears
 to be smooth
- Line Style choose type of line used for plot
- Symbol Style choose type of symbol displayed in plot
- Fill Style choose the fill style for the objects in the plot that use it
- Line Colors sets the color sequence for multiple lines displayed on the chart
- Fill Colors sets the color sequence for sequential symbols or gradient used for plot display
- Number of Color bands when set to 0, the gradient will be a smooth graduation between values, rather than banded, where a single color is shown for a range of values
- Relative Bar Width sets the width of a bar as a proportion of available space [0 1], determined by
 the maximum size bar that can be displayed without overlapping any adjacent bars of the same variable
 (if other variables appear between bars of this variable, making the bar wider may overlap those intervening variables)
- Relative Bar Offset sets the start position of a bar, proportional to the minimum point where the bar could be placed to the maximum point where the bar could be placed [0 - 1], with the maximum size based on the same criteria as Relative Bar Width
- Show Linear Interpolation of Lines causes the ends of a line plot to extend to the edge of the chart, when the plot is not against a discrete axis
- Symbol Outline Colors sets the color used to outline displayed symbols
- Symbol Size sets the size of displayed symbols in pixels [1 16]
- Line Width sets the width of lines (including lines outlining bars), in pixels [1 10]

Note

In most cases, properties are not shown if they are not applicable to the item selected. However, Show Linear Interpolation of Lines is displayed whether the axis is discrete or continuous, but it only applies in cases of continuous axes. The option will have no effect if you set it when the plot is displayed on a discrete axis.

Chart Properties

Most charts have a few general properties that appear under a **Chart** entry in the **Properties** view when you click on the background of a chart and select **Edit Properties**. When plot line rendering is controlled generally rather than per variable (as for spider charts, for example), the **Style** display properties will also appear when you edit the chart properties. Chart properties may include:

- Title chart title that appears in the **Chart** view title bar
- Chart Type allows some charts to be displayed as another chart type
- Display Percentages turns on percentage values on pie charts

Chart Zoom, Pan, and Rotate

You can manipulate the display of a 3D chart using the zoom, pan, and rotate features.

- Zoom by using the mouse wheel or Shift + middle mouse button
- Box zoom by using the right mouse button
- Pan by using the Ctrl + middle mouse button
- · Rotate by using the left or middle mouse buttons

Pan and rotate by holding down the appropriate key sequence and moving the mouse.

There are several ways that you can enlarge or shrink portions of the chart in order to view more or less detail. To zoom the entire chart, click on the chart and use the scroll wheel or Shift + middle mouse button to magnify or shrink the chart. The chart expands or shrinks as you roll the wheel (or move the mouse) toward you or away from you, remaining centered in the view pane. The chart can be panned or rotated when it is magnified.

To magnify a particular area of a chart, right-click and drag a box top to bottom over the area you want to view. The contents of the box you draw will be magnified to fill the chart view, so the smaller the box you draw, the closer the magnification on a particular area.

If you right click and draw a box bottom to top over the chart, the chart will shrink to roughly correspond to the size of the box that you have drawn, so a very small box will give you a tiny chart.

Using the Triad

On three dimensional charts, the *triad* appears in the lower left corner of the chart view, showing the orientation of the three axes for the current view of the chart. The x axis is red, the y axis is green, and the z axis is blue. There is a light blue ball in the triad that indicates the orientation ISO z axis up position of the chart. If you click on this ball it will set your chart view to be ISO z axis up, fit to window.

If you move your cursor around the triad, you will see a yellow arrow appear that shows the direction that corresponds to the position of your cursor (+x, -x, +y, -y, +z, -z). If you click on the arrow, it changes your chart view so that the chart axis indicated by the arrow is facing out.

There are several shortcut keys that can be used when you are viewing a 3D chart:

- f fits the chart to the window
- x displays the x+ view, fit to window
- y displays the y+ view, fit to window
- z displays the z+ view, fit to window
- i displays the chart in the ISO z axis up position, fit to window

Saving a Chart

You can save the chart that you are viewing as a graphic. To do so, right click on the background of the chart and select Save Image As. In the dialog that appears, you will see a small image of the chart, and can select the **Size** (resolution) that will be used when saving the chart. Click on the ellipsis

button and navigate to the folder where you want to save the file. Enter a filename. You can select either .png or .bmp as the graphic file type. Click **Save** to select that file path as your save location. Click **OK** to save the file to the location that you selected, with the resolution that you've selected.

Working with Project Reports

ANSYS Workbench project reports allow you to generate a visual "snapshot" of your project. The contents and organization of the report reflect the layout of the **Project Schematic**, with sections for global project information, analysis system information, system cell information, and where applicable, content provided by applications in the project.

Note

At this time, DesignXplorer and CFD-Post are the only applications that provide the project report with detailed report content. Content for other applications is limited to the data visible at the project level (for example, in the **Properties** view for the associated cell in the **Project Schematic**).

This section addresses the following topics:

Configuring Project Reports (p. 86) Generating Project Reports (p. 87) Report Content for Projects with Design Points (p. 87) Editing Project Reports (p. 87)

Configuring Project Reports

Setting Project Report Options

You can specify report settings in the Workbench **Options** dialog.

- 1. In Workbench, select **Tools > Options**.
- 2. In the **Options** dialog, expand the **Project Reporting** option in the navigation tree.
- 3. Select the **After exporting report, automatically open in default browser** check box to launch your default browser and load the report immediately upon generation. If you do not select this option, you will need to navigate to the report file and open it manually.

Including CFD-Post Data in the Project Report

To include CFD-Post data in your project report:

- On the Project Schematic, right-click the Results cell for a ANSYS CFD analysis system or Results system and select Properties.
- 2. In the **Properties** view under **Project Reporting**, select the **Publish Report** check box.

3. Repeat for each **Results** cell to be included in the project report.

Note

Only 2D content such as graphs and figures are supported in the project report. If interactive 3D content exists, it will be displayed in 2D format.

Generating Project Reports

The content and status of the project determine the content of the report. The project report reflects the current state of the project at the time the report is generated.

To generate a project report, select **File > Export Report**.

Report Content for Projects with Design Points

If the project includes parameters and report content has been provided by an application in the project during design point update, then the **Design Points** table will contain links to sub-reports for each design point. Detailed report content for each design point can be accessed via a hyperlink in the **Report** column of the **Design Points** table in the project report. Clicking on the link will open a sub-report that contains the application-specific content for that design point (if available).

Editing Project Reports

Once you have generated a project report, you can edit its contents as needed.

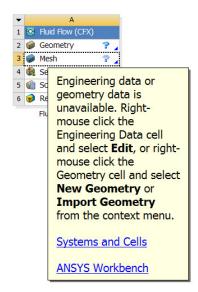
- 1. Open the report file with an HTML-adapted editor (such as Microsoft Word) by right-clicking the file and selecting the **Open with** menu option.
- 2. Edit the report contents and formatting as needed.
- 3. Save the file in the desired file format(s) to a location outside Workbench.

Using Help

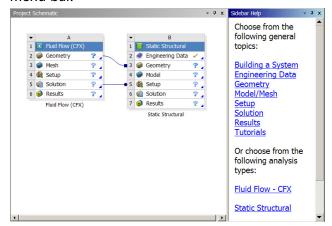
ANSYS Workbench offers three levels of help:

- Quick Help (p. 87)
- Sidebar Help (p. 88)
- Online Help (p. 88)

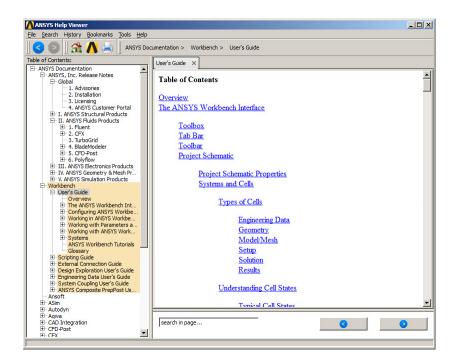
Quick Help Quick help is available for most cells in a system. Click the blue arrow in the bottom right corner of the cell to see a brief help panel on that cell. Quick help is generally state-sensitive; as the state of a cell changes, the content of the quick help panel will update to match. From quick help, you can also access related help topics in the online help system.



Sidebar Help Context-sensitive help is available at any time by clicking **F1**. The **Sidebar Help** view will be displayed on the right side of the screen. The content of this help panel is determined by the portion of the interface that has focus (i.e., where the mouse was last clicked). If no particular area has focus, you will see a general GUI overview topic. If the **Project Schematic** has focus but no systems are defined, you will see a **Getting Started** topic. If the **Project Schematic** has focus and one or more systems have been defined, you will see links to those specific system types, as well as links to general topics. You can also access the **Sidebar Help** view by choosing **Help** > **Show Sidebar Help** from the menu bar.



Online Help Online help is available from the ANSYS Workbench **Help** menu, or from any of the links in the quick help or the **Sidebar Help** view. Online help provides a comprehensive discussion of all ANSYS Workbench features and capabilities, and includes a full search capability.



Troubleshooting

This section lists problems and error messages that you may encounter while running ANSYS Workbench. After each situation description or error message is the user action required to correct the problem. This section does not include troubleshooting for data-integrated applications. For troubleshooting information for data-integrated applications, please see the help for the specific application.

For additional troubleshooting information on native applications, please see the following:

- DesignXplorer Troubleshooting
- DesignModeler Frequently Asked Questions

You can find additional FAQs on the *Product Documentation* section of the *ANSYS Customer Portal* at (https://www1.ansys.com/customer/).

Problem Situations

During setup, if you encounter any errors containing the text "0x8000FFFF", you will need to install the required installation prerequisites. Run the installation launcher (setup.exe) and choose **Install Required Prerequisites**.

CAD System Plug-In Menus Do Not Appear for NX or Creo Parametric ANSYS Workbench on Windows platforms will append its information to an existing customization file for NX and/or Creo Parametric. If no customization file exists, ANSYS Workbench will create a file. For NX, ANSYS Workbench looks for the custom_dirs.dat file in the directory specified via the UGII_CUSTOM_DIRECTORY_FILE environment variable. For Creo Parametric, ANSYS Workbench looks for the config.pro file in the %HOMEDRIVE%%HOMEPATH% directory. In addition, during setup of the Creo Parametric Geometry Interface, ANSYS Workbench will also append its information to the config.pro file located in the Creo Parametric installation path, under the \text directory (e.g., Proewildfire2\text\config.pro).

If ANSYS Workbench encounters a read-only file, it will not be able to write the necessary information to the file. In this case, you will need to revise the permissions on the file and manually add the appro-

priate ANSYS Workbench-specific information in order for the ANSYS menu to appear in NX or Creo Parametric.

Script Errors When Running ANSYS Workbench If you encounter script errors such as "Error: Unable to create object microsoft.XMLDOM," you may need to install the latest version of Microsoft's MSXML. Please visit Microsoft's web site at http://www.microsoft.com/downloads/details.aspx?FamilyID=993c0bcf-3bcf-4009-be21-27e85e1857b1&DisplayLang=en for more information on downloading and installing MSXML.

Charts Do Not Appear If you do not see the appropriate charts created after a design point run or other updates (such as response surface updates), reset the tab by going to **View** > **Reset Workspace**.

Applications Do Not Start After Crash On Linux, if ANSYS Workbench applications will not launch after an abnormal exit such as a crash, run the following utility:

\$INSTALL/v150/aisol/wbcleanup

Wobbly Desktop Effect Causes Crashes The Wobbly desktop effect on Linux may cause ANSYS Workbench or its applications to crash. This effect is on by default on some Linux platforms. Make sure the effect is turned off.

System Freezes After Certain GUI Operations There are known issues involving UI operations if you are using the KDE 3.5 desktop environment on Linux systems when running ANSYS Workbench products. If using KDE 3.5 and you open a drop-down list in the Details View, you must select an entry from the list before performing any other UI operation or the product may hang. As an alternative, use the GNOME desktop environment.

Startup or Graphics Problems

To minimize graphics problems, always verify that you are running the latest graphics drivers provided by your computer's hardware manufacturer. If you are not, you may see the following message:

***An error occurred while setting up the graphics window. Please ensure that you have the latest drivers from your graphics card manufacturer. If the error persists, you may need to decrease the graphics acceleration. For more information, please see the Troubleshooting section in the ANSYS Workbench help.

Linux If you are running ANSYS Workbench on Linux and experience problems at startup or with the GUI or graphics displaying correctly, and you are running in accelerated graphics mode, you may need to relaunch ANSYS Workbench using the <code>-oglmesa</code> flag to activate software rendering:

```
runwb2 -oglmesa
```

If ANSYS Workbench detects that graphics problems are causing crashes, it will automatically switch to software rendering. ANSYS Workbench also will use software rending mode by default when running on a remote display, or on a local display if the hardware does not appear to be accelerated.

To revert to accelerated graphics mode, launch ANSYS Workbench using the -oglhw flag:

```
runwb2 -oglhw
```

If you are running under Exceed3D, try the following settings if you are having graphics problems:

- Turn off the graphics (hardware) acceleration option in Exceed3D options.
- If graphics acceleration is on, turn on the GLX 1.3 option.

Any version of Exceed that does not have the GLX 1.3 option is unlikely to function correctly with graphics acceleration.

Windows If you experience graphics issues on Windows systems, you can find the details of your graphics card and the driver that is currently installed by running **Start> Run** and typing the following into the **Open** field:

dxdiag

Select the **Display** tab and review your graphics card information. You can then contact the vendor or visit the vendor's website for details of the latest graphics drivers available for your specific graphics card.

You may also have to adjust the hardware acceleration. To adjust the hardware acceleration, go to the Control Panel and choose **Display> Settings> Advanced> Troubleshoot** or your operating system's equivalent. The hardware acceleration slider should then be visible. You can also choose **Display> Settings> Troubleshoot** and use the troubleshooting tool to guide you to the Hardware Acceleration panel. Try turning hardware acceleration off by dragging the slider to **None**. Try the software again; if the graphics problems are resolved, then gradually increase Hardware Acceleration as far as you can before the software fails again.

Limitations

Selecting a preference in the **Details** view while another property is being edited may result in a system failure or freeze on Linux 64-bit operating systems when using KDE (K Desktop Environment) 3.5.

Error Messages

***Unable to connect to Solver Manager.

Another application might be using the Solver Manager port (10002 by default). Try changing the port number by editing the Ansys. SolverManager.exe.config file located in the installation directory at \AISOL\Bin\platform>.

If you are getting the "Unable to connect to Solver Manager" error message or are having difficulty launching other applications/editors, it is also possible that the Windows hosts file has been corrupted. Please make sure that localhost is specified in the Windows <os drive>:\Windows\sys-tem32\drivers\etc\hosts file.

***FATAL

***Parallel capability is not valid for this product

If you see this message in the Mechanical APDL output window when attempting to run Mechanical APDL with an ANSYS LS-DYNA license (commercial or academic) from ANSYS Workbench, set the number of processors for Mechanical APDL to 1 (**Tools> Options> Mechanical APDL**). You will then be able to run Mechanical APDL and solve an ANSYS LS-DYNA analysis.

Warning at File: myxml, line 1, col 40, Encoding (utf-16, from XMLDecl or manually set) contradicts the auto-sensed encoding, ignoring it.

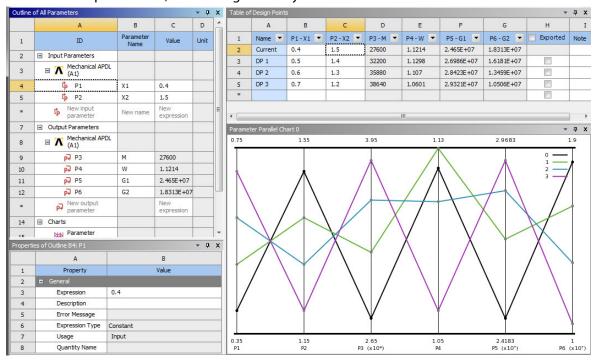
The message above will sometimes be displayed in the RSM log; you can ignore it.

Release 15.0 - © SAS IP, Inc. Al	Trights reserved - Conta	inc proprietary and	confidential inf	ormation
nelease 13.0 - @ 3A3 IF, IIIC. AI	i ngnis reserveu Contu	iiris proprietary ana	Commuentiarini	omination
-11	NSYS. Inc. and its subsid	l:		
OI A	an sy s. inc. ana its subsia	iaries ana attiliates.		

Working with Parameters and Design Points

Parameters and *design points* are native features of ANSYS Workbench. From with the various simulation applications, you can promote key simulation properties to parameters. You can then manipulate parameters at the project level, using them to investigate a series of design alternatives. A single set of parameter values representing one design alternative is called a design point. A set of design points can be created in tabular form and run automatically to perform a what-if study.

ANSYS Workbench stores and coordinates all of the parameters and design points that are produced or consumed by components of the current project. For the most part, you will work with defined parameters and design points in two different tabs: the **Parameter Set** tab and the **Parameters** tab. Parameters and design points can also be used by ANSYS DesignXplorer for different types of automated design exploration studies, which, for DesignXplorer, include Parameters Correlation, Response Surface, Goal Driven Optimization, and Six Sigma Analysis.



Related Topics:

The Parameter Set Tab or Parameters Tab Parameters Design Points

The Parameter Set Tab or Parameters Tab

At the project level, for the most part you will work with parameters and design points in two different workspaces, the **Parameter Set** tab and/or the **Parameters** tab. The **Parameter Set** tab includes all of the parameters and design points in all of the systems in the project, while a **Parameters** tab includes only the parameters and design points in the selected system.

To access the **Set** tab or a **Parameters** tab, double-click the **Parameter Set** bar or a **Parameters** cell. Alternatively, you can right-click either one and select **Edit** from the context menu. For information on defining parameters, refer to the documentation for each application with which you are working.

Both tabs include an **Outline** view, a **Properties** view, a **Table** view, and a **Chart** view that display parameter and design point data in different formats. These views allow you to see the interaction of parameters between applications, providing you with greater flexibility and the ability to run optimization and/or what-if scenarios.

Related Topics:

Working with Parameters in the Parameter Set Tab or Parameters Tab Working with Design Points in the Parameter Set or Parameters Tab

Working with Parameters in the Parameter Set Tab or Parameters Tab

The process of working with parameters begins when you define a parameter in an application, such as your CAD system, Mechanical, or Fluent. (For information on defining parameters, refer to the documentation for each application with which you are working.)

ANSYS Workbench recognizes parameters defined in the individual applications and exposes them in a **Parameter Set** that can be shared by multiple systems. The **Parameter Set** bar is the visual representation of the project's full parameter set and allows you to open the **Parameter Set** tab. The **Parameter Set** tab includes all of the parameters defined for all of the systems in your project, with each parameter identified by their system of origin.

Additionally, each system with parameters has a **Parameters** cell, which allows you to open the corresponding **Parameters** tab (i.e., multi-system projects can have multiple **Parameters** tabs). The **Parameters** tab for a given system includes all of the parameters defined for that system.

Each of these tabs has an **Outline** view and a **Properties** view which allow you to view and/or work with your parameters.

Outline View

The **Outline** view lists the parameters, grouping them into **Input Parameters** (parameters that affect the definition of the data model) and **Output Parameters** (analysis results that are quantities of interest for the design).

Note

In a **Parameters** tab, the title in the **Outline** view header incorporates the ID of the corresponding **Parameters** cell to indicate the source of the parameters. In the **Parameter Set** tab, the title is **Outline of All Parameters**.

For each parameter, the **Outline** view shows an ID, name, current value, and unit system. You can edit most of these properties; the exceptions are the units (for parameters with quantity values) and the parameter ID. When you select a parameter, details for that parameter are shown in the **Properties** view.

You can also add new parameters in the **Outline** view. For parameters created in this way, you can assign a value, but not an expression. (To add an expression, select the newly created parameter and add the expression and quantity name in the **Properties** view.)

Note

Opening a file from a previous release can result in an *unlinked* parameter (i.e., a parameter associated with a property that existed in the previous release but does not exist in the current one). Unlinked parameters will be labeled as such in the **Outline** view **Value** column. To delete an unlinked parameter, right-click and select **Delete Selected Unlinked Parameters**.

Properties View

The **Properties** view displays information for the object (either a parameter or a chart) selected in the **Outline** view. From this view, you can perform the following operations:

- enter or change the value of input parameters
- specify details for a parameters created in the **Outline** view
- · delete existing user-defined parameters
- change parameter names
- · edit parameter descriptions
- enter or edit parameter expressions
- link an input parameter to an output parameter by editing its expression

When working with expressions, once the value quantity name (e.g., Area) is determined from the unit of the value, you can only modify it by changing the parameter's **Quantity Name** setting. You cannot change the expression (e.g., from Area to Volume) without first changing the quantity name value.

After making any changes to parameter definitions, perform an Update operation to run the needed updates and return the values of the output parameters. Note that an Update operation can be lengthy, depending on the analysis details.

For more information on working with parameters, see Parameters (p. 96).

Working with Design Points in the Parameter Set or Parameters Tab

A **design point** is a set of input parameter values and corresponding output parameter values associated with an individual parameterized project definition. Each design point is a single set of parameter values representing one design alternative. Basically, you can think of a design point as a snapshot of your design given a set of parameter values, where output parameter values have been calculated by an update of the project. Design points allow you to perform what-if studies and are also used by ANSYS DesignXplorer for design exploration.

Most of your work with design points will occur in the **Table** view of the **Parameter Set** tab and/or a **Parameters** tab. In both tabs, the **Table** view provides a tabular view called the **Table of Design Points**. From the **Table of Design Points** view, you can view design points and perform the following operations:

From the **Table of Design Points** view, you can perform the following operations:

- create new design points
- · duplicate existing design points
- · enter or change the input parameter values for design points
- delete existing design points (except for the Current design point)
- update design points, either locally or via Remote Solve Manager
- activate and export design points
- reserve licenses for design point updates
- · specify the design point update order

For more information on working with design points, see Design Points (p. 101).

Parameters

A **parameter** is an entity that is linked to a data model property within an application. You can have input and output parameters. Parameter values can be numeric or non-numeric (string or Boolean). Non-numeric parameters are ignored for charting purposes. An input parameter can be modified at the project level and drives a change within the data model. The value of an output parameter is set by the application, based on the current results or state.

Input parameters are parameters that define the geometry or inputs to the analysis for the system under investigation. Input parameters have predefined values or ranges that may be changed. These include (and are not limited to) CAD parameters, analysis parameters, and DesignModeler parameters. CAD and DesignModeler input parameters might include length, radius, etc.; analysis input parameters might include pressure, material properties, sheet thickness, etc.

Output parameters are parameters that result from the geometry or are the response outputs from the analysis. These include (but are not limited to) volume, mass, frequency, stress, velocities, pressures, forces, heat flux, and so forth.

Custom Parameters are input or output parameters you have created and that are defined by an expression. For more information, see Custom Parameters (p. 96).

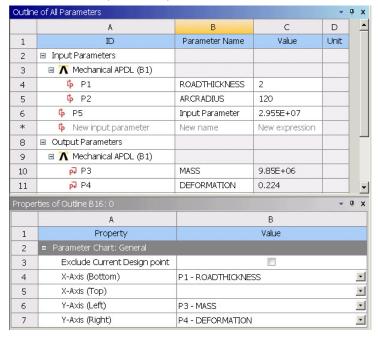
All parameters have a type, preferably a quantity with a quantity name. The quantity name is used to define preferred and available units for the quantity. If the value is not a quantity or does not have a quantity name set, it will be displayed as unitless throughout the GUI.

Parameters can be added to or deleted from the current project, but this may set the existing design points and Design Exploration systems to an out-of-date state and could result in several hours of recalculation time, depending on the project. Be aware that deleting a parameter referenced in the expression of another parameter will invalidate the driven data model, resulting in an error.

Custom Parameters

At the project level, you can create custom parameters that are not directly associated to a data model property. They can be custom input or custom output parameters that can be defined by a constant value (e.g., "12.5 [cm]" or "sin(pi/2)"), or they can be derived parameters, defined by an expression of other parameters (e.g., "P2+3*P3").

To create a derived parameter, enter the expression statement in the **Expression** field of the **Properties of Outline** view or in the **Value** field of the **Outline of Parameters** view. Both derived parameters and constant values can be added in the **Outline of Parameters** view, but once added, derived parameters become read-only in the **Outline** view. You must use the **Properties of Outline** view to edit them. Constant values, however, can be edited in the **Outline of Parameters** view.



If the expression defining the custom parameter results in a quantity, it will infer its quantity name from the value produced by the expression evaluation. As a consequence, if the expression is a sum of multiple terms, every term needs to be made of the same quantity. For example, if the resulting quantity is 3.4[m^2], the new custom parameter will take on the quantity name "Area." In some cases, more than one quantity name may be valid for the expression. In these cases, the parameter quantity name will not be set, but can be selected from a list in the parameter **Properties** view. Once the value quantity name (e.g., Area) is determined from the unit of the value, it can only be subsequently changed by changing the parameter's "value quantity name" property setting. You cannot change the expression (e.g., from area to volume) without changing the value quantity name.

The expression defined for a custom boolean parameter can be the Python values **True** or **False**, or a Python logical expression, such as "P1 > P2" or "P1 == 10 and P2 == 10". For a parameterized boolean parameter, you can select **True/False** from the drop-down in the **Value** column.

The expression defined for a custom string parameter must be quoted with single or double quotes, i.e., 'string value' or "string value".

Output to Input Chaining of Parameters

You can chain output to input parameters to allow an input to be driven directly from the current value of an output, provided that the chaining does not create a circular dependency. To chain parameters, change the input parameter definition to an expression involving other parameters. To create a derived variable, first insert an expression and then edit the **Expression** field in the **Properties of Outline** view.

Expressions, Quantities, and Units

The ANSYS Workbench expression parser supports standard math functions and operators, as well as units for quantities. Dimensional quantities are defined in units which are a combination of one or more separate units.

ANSYS Workbench expression and mathematical function evaluation is based on the Python 2.6 programming language (www.python.org) and inherits some behavior as described here. All Python numeric and function capabilities can be used. For example, Python provides support for octal (base 8) and hexadecimal (base 16) numbers. To tell Python that a number should be treated as an octal numeric literal, simply append a zero to the front. Appending a zero and an x to the front of a number tells Python to treat the number as a hexadecimal numeric literal.

Do not start expressions with an "=" operator. To define a derived parameter P5 such that P5=P2+3*P3, set the expression to P2+3*P3, given that P2 and P3 are existing parameters.

Expressions that involve quantities must be dimensionally consistent. The + and - operators require that the two operands have compatible units. For example, you cannot add an Area parameter to a Length parameter; both units must be Length or both units must be Area. The * and / operators do not have this limitation. They allow one operand to be a quantity with a unit and the other operand to be a dimensionless factor. Or, they allow both operands to be quantities with units where the result is a different quantity type. For example, Length/Time results in a quantity with a Velocity unit.

Expressions support a number of intrinsic functions, outlined below. These functions support both standard numeric values and quantities as arguments.

abs(arg)	cosh(arg)	log10(arg)	sin(arg)
acos(arg)	exp(arg)	max(arg list)	sinh(arg)
asin(arg)	fabs(arg)	min(arg list)	sqrt(arg)
atan(arg)	floor(arg)	nint(arg)	tan(arg)
atan2(arg1,arg2)	int(arg)	pow(value, exponent)	tanh(arg)
ceil(arg)	log(arg)	round(arg)	PI, pi – constant
cos(arg)	loge(arg)	sign(arg)	E, e - constant

The arguments for trigonometric functions are evaluated as follows:

- If the argument is a number (i.e., real or integer), then the argument is evaluated as radians.
- If the argument is a quantity (i.e., has value and units), then the argument must be of type 'Angle', and evaluation is based on the supplied units (i.e., [deg] or [rad]).

You can include units, assuming the unit makes sense in context of the expression. For example, P2+3[mm]*P3 is valid if mm is a valid unit expression for P2 and P3 (e.g., if P2 is torque and P3 is force, or if P2 is area and P3 is length).

The project unit system is used to evaluate the expressions. For temperatures, absolute temperature values are used in expression evaluation. All quantity values in an expression are converted to the project unit system.

The general units syntax in ANSYS Workbench is defined as [multiplier|unit|^power], where multiplier is a multiplying quantity or its abbreviation (e.g., mega (M), pico (p), etc.), unit is the unit string or abbreviation (e.g., gram (g), pound (lb) foot (ft), meter (m), etc.), and power is the power to which the

unit is raised. See Table 2: Unit Multipliers (p. 99) and Table 3: Example Quantities and Units (p. 100) for examples of multipliers and commonly used units.

When typing units in an expression, the units must be enclosed by square braces [...]. You will usually not see the braces when selecting units from a list of commonly used units. In general, units declarations must obey the following rules:

- A units string consists of one or more units quantities, each with an optional multiplier and optional power. Each separate units quantity is separated by one or more spaces.
- Abbreviations for multipliers and unit names are typically used, but full names are also supported.
- Powers are denoted by the ^ (caret) symbol. A power of 1 is assumed if no power is given. A negative power is typically used for unit division (i.e., [kg m^-3] corresponds to kilograms per cubic meter).
- If you enter units that are inconsistent with the physical quantity being described, then an expression error will result.
- Units do not have to be given in terms of the fundamental units (mass, length, time, temperature, angle, and solid angle). For instance, Pa (Pascals) and J (Joules) are both acceptable as parts of unit strings.
- Units strings are case sensitive; for example, Kg and KG are both invalid parts of units strings; kg is correct.

Caution

English US expression evaluations that involve temperatures are a special case. For the unit conversion of a specific temperature value, 1 degC = 274.15 K. However, the unit conversion for a temperature interval (delta T) is 1 degC = 1 K. The expression evaluator will take any temperature value and treat it as a specific temperature (not a temperature interval) by converting it to the absolute unit of the project unit system (either K or R). If the intent is to perform the evaluation in terms of temperature intervals, you need to start with temperatures in absolute units.

Once the value quantity name (e.g., Area) is determined from the unit of the value, it can only be subsequently changed by changing the parameter's "value quantity name" property setting. You cannot change the expression (e.g., from area to volume) without changing the value quantity name. Automatic unit conversion is only done when the quantity name is known.

Table 2: Unit Multipliers

Multiplier Name	Multiplier Value	Multiplier Abbre- viation
exa	10 ¹⁸	E
peta	10 ¹⁵	Р
tera	10 ¹²	Т
giga	10 ⁹	G
mega	10 ⁶	M
kilo	10 ³	k
hecto	10 ²	h
deca	10 ¹	da

Multiplier Name	Multiplier Value	Multiplier Abbre- viation
deci	10 ⁻¹	d
centi	10 ⁻²	С
milli	10 ⁻³	m
micro	10 ⁻⁶	u
nano	10 ⁻⁹	n
pico	10 ⁻¹²	р
femto	10 ⁻¹⁵	f
atto	10 ⁻¹⁸	a

Table 3: Example Quantities and Units

Quantity	Dimensionality	Example Units
Acceleration	Length Time^-	m s^-2
	2	ft s^-2
		in s^-2
		um ms^-2
Current	Current	А
		mA
		рА
Density	Mass Length^-	kg m^-3
	3	g cm^-3
		lb ft^-3
		slug in^-3
		slinch in^-
		3
Electric Charge	Current Time	A s
		coulomb
		pA s
Energy	Mass Length^2	J
	Time^-2	BTU
		erg
		lbf ft
		slug in^2
		s^-2
Force	Mass Length	dyne
	Time^-2	N
		pdl
		lbf
		slug in s^-
		2
Length	Length	m
		cm
		foot
		in
		mm

Quantity	Dimensionality	Example Units
		micron
		ft
		um
		yard
Pressure	Mass Length^-	Pa
	1 Time^-2	MPa
		N m^-2
		bar
		torr
		mm Hg
		psi
		psf
		atm
		dyne
		cm^-2
Power	Mass Length^2	W
	Time^-3	BTU s^-1
		HP
		erg s^-1
		lbf ft s^-1
Temperature	Temperature	К
		С
		R
		F

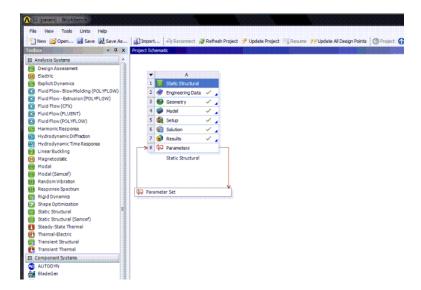
Design Points

A **design point** is a set of input parameter values and corresponding output parameter values associated with an individual parameterized project definition. Each design point is a single set of parameter values representing one design alternative. Basically, you can think of a design point as a snapshot of your design given a set of parameter values, where output parameter values have been calculated by an update of the project.

The project that you interact with via the **Project Schematic** is always the Current design point (DP0). You cannot rename or delete Current (DP0), but you can add new design points via either of the following methods:

- Duplicate an existing design point. Right-click a design point in the **Table of Design Points** and select **Duplicate**. When you duplicate a design point, all parameters, parameter values, design point states, and design point **Export** settings are copied. You can then modify any of these values as required.
- Create a new design point. Enter a parametric value in a cell in the bottom row of the **Table of Design Points**, as shown in the following animation.

The following Show-Me Animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



You can check out licenses to ensure the appropriate licenses are available for a design point update. For more information, see Updating Design Points (p. 103).

Changes made to a project with up-to-date design points may cause all existing design points to go out of date, if the change is related to the parametric study being performed. Any change not relevant to the parametric study, such as adding a standalone system or making a change downstream of the study, should not cause design points to go out of date. For changes that invalidate the **Table of Design Points**, you will need to re-update the project, which could require significant time and/or computing resources. You should always save your project after updating all design points and before further modifying the project.

In most cases, a design point update operation applies only to cells affected by parameter changes; cells without associated parameters or cells whose associated parameters did not change are not updated.

Note

The exception is External Connection and CFD-Post systems. For these, you can specify that some cells will always be updated with a design point update operation, even when parameter values are not affected. To do so, open the cell **Properties** view and select **Always Include in Design Point Update**.

To update all project cells regardless of whether the parameters associated with those cells have changed, you must select **Update Project**. Cell states will reflect this behavior.

Related Topics:

Updating Design Points
Design Point Update Order
Activating and Exporting Design Points
Updating Design Points via Remote Solve Manager (RSM)
Reserving Licenses for a Design Point Update
Design Point Update Data
Design Point States

Updating Design Points

Given a set of input parameter values for a design point, output parameter values are calculated by updating the design point or the set of design points. Updating design points will only update solution data where output parameters have been defined.

Note

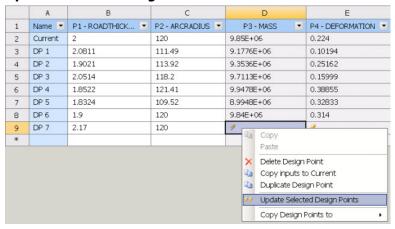
For External Connection and CFD-Post systems, it is possible to specify that a cell will always be updated with a design point update operation, even when the update will not affect parameter values. To do so, open the cell properties and select the **Always Include in Design Point Update** check box.

You have several options to update design points:

Update One Design Point

You can choose to update just one design point:

In the **Table of Design Points** in the **Parameters** tab, right-click on the desired design point and select **Update Selected Design Points** from the context menu.



Note

Selecting **Update Project** from the **Toolbar** or the **Project Schematic** context menu updates only the Current design point (DP0) and does not update any other design points.

Update All Design Points

You can choose **Update All Design Points** from the **Toolbar** or the **Project Schematic** context menu. This option will update all design points in the project.

>> Update All Design Points

Update a Selected Set of Design Points

You can choose to update a set of selected design points:

- 1. In the **Table of Design Points** in the **Parameters** tab, hold the **Ctrl** key and left-mouse click on each of the design points to be updated.
- 2. Release the **Ctrl** key and then right-click one of the selected design points.
- 3. Select **Update Selected Design Points** from the context menu. Settings specified via the **Design Point Update Order** setting described below will apply.

If you are updating multiple design points and the update for one of the design points fails, the **Update**

Failed, Update Required () icon will show for the failed design point, but the next design point update will begin immediately. In this case, the Messages view will open automatically, showing the error message for the design point(s) that failed. A failure summary dialog will appear when the update process has completed; if you see this dialog, be sure to review the error messages in the Messages view for details on which design point(s) failed.

Design Point Update Order

By default, design points are solved in the order in which they appear in the **Table of Design Points**. As each design point is updated, it begins from the parameter values associated with DP0 (Current). When updating design points, it may be more efficient to update design points starting from parameter values from the previous design point, rather than starting from DP0 each time. For example, If DP2 has the same geometry as DP1, but both of these differ from DP0, then updating design points in order would save the computational cost of updating both the geometry AND the mesh for the DP2.

You can specify how the design points will be updated from the **Properties** view of the **Parameter Set** bar. Right-click the **Parameter Set** bar and select **Properties** to view the **Design Point Update Order** settings:

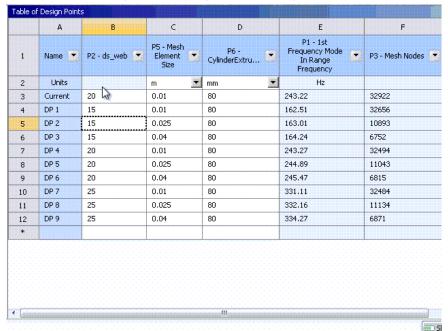
- Update from Current will cause each design point to be updated starting from DP0 (the default).
- **Update Design Points in Order** will cause each design point to be updated starting from the previous design point.

You can also specify in which order the design points will be updated so that the efficiency of the operation is improved. For instance, if several design points use the same geometry parameter values, it is more efficient to process them together in order to update the geometry only once.

In the table of design points, the **Update Order** column shows the order number for each design point. To view the **Update Order** column, right-click in the table and select **Show Update Order**. By default, it matches the order of appearance in the table, but you can modify it in different ways:

- By editing the values in the **Update Order** column.
- By sorting the table by one or several columns and then right-clicking in the table and selecting Set Update
 Order by Row. This option will regenerate the Update Order for each design point to match the sorting
 of the table.
- Automatically, by right-clicking in the table and selecting Optimize Update Order. This option will analyze
 the parameter dependencies in the project and scan the parameter values across all design points in order
 to determine an optimal order of update. This operation modifies the Update Order value for each design
 point and refreshes the table.

The following Show-Me Animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



Activating and Exporting Design Points

Activating Design Points

ANSYS Workbench does not save the calculated data for each design point other than "Current" unless you export that design point. Only the actual parameter values are saved. Therefore, when you activate a different design point, you need to recalculate the results. You can activate a different design point by using the **Copy inputs to Current** option on the design point's context menu. When you activate different design points, any editors that are open may close.

Exporting Design Points to New Projects

To save all data for a design point other than Current, you can export the design point as follows:

- 1. Select the **Exported** check box for the desired design point in the **Table of Design Points** view. Alternatively, you can select the check box in the **Exported** header cell to select all design points for export.
- 2. Specify the design points to be updated via one of the following methods:
 - Right-click an individual design point and select Update Selected Design Points from the context menu.
 - Click the **Update All Design Points** button from the toolbar.

Either of these actions will write out separate projects (one for each design point selected) using the values of the selected design point.

The projects resulting from the export will be fully independent and up-to-date projects named $\projectname>_dpn.wbpj$, located as siblings in the same directory as the main project. Your parent project must be saved before you can save any subsequent design points. If the design point

fails to export, the files for that design point will remain in the project_files\dpn directory until you delete them manually or attempt to update the design point again.

To export a design point, you must mark it for export before updating it. If a design point is already up-to-date and is then marked for export, you must update the design point again in order to export it. When the design point is updated, Workbench performs a full update to ensure that, when possible, all of the cells and systems in the exported project are up-to-date.

If you change a parameter in an exported design point and then run an update, the existing exported project files will be overwritten. However, if you first deselect the **Exported** check box, then change the design point, and then run an update, the exported project will not be overwritten but will no longer be consistent with the data in the **Parameter** table.

Note

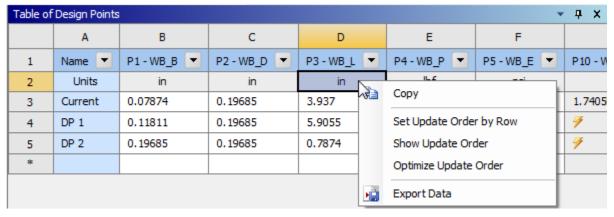
Be aware that Design Exploration data is not a part of design points and therefore is not exported. Design Exploration is a consumer of design points but does not define design points and is not involved when a design point is updated, or by extension, exported.

Exporting Design Point Values to an ASCII File

You can export the design point values in the **Table of Design Points** to an ASCII file, which can then be used by other programs for further processing. This file is formatted according to DesignXplorer's "Extended CSV File Format," which is primarily the Comma-Separated Values standard format (file extension .CSV) with the addition of several nonstandard formatting conventions.

To export design point values to an ASCII file, right-click on a cell in the **Table of Design Points** and select the **Export Data** menu option. The parameter values for each design point in the table will be exported to a .CSV file. The values are always exported in units as defined in Workbench (i.e., as when the **Display Values as Defined** option is selected under the **Units** menu).

If you export design point values from the **Table of Design Points** in the example above, the following file will be generated:



```
#
# 10/1/2012 10:38:01 AM
# The parameters defined in the project are:
# P1 - WB_B [mm], P2 - WB_D [mm], P3 - WB_L [mm], P4 - WB_P [N], P5
- WB_E [MPa], P10 - WB_SIG [MPa], P8 - WB_DIS [mm], P9 - WB_BUCK [N]
#
```

```
# The following header line defines the name of the columns by refer-
ence to the parameters.
Name, P1, P2, P3, P4, P5, P10, P8, P9
DP 0, 2, 5, 100, 1000, 200000, 12000, 80, 1028.91145833333
DP 1, 3, 5, 150, 1000, 200000, , ,
DP 2, 5, 5, 20, 1000, 200000, , ,
```

For details on the file formatting, see Extended CSV File Format in the Design Exploration User's Guide.

Updating Design Points via Remote Solve Manager (RSM)

As an alternative to updating design points locally, you can choose to update your design points by submitting them to Remote Solve Manager (RSM).

Important

- If you are sending design point update jobs to a remote computing cluster, note that the license preferences set for your local machine (the RSM Client) may not be the same as the license preferences set for the remote cluster machine (the Compute Server). In this case, the Compute Server license preferences will be used for all jobs. For more information, see Establishing User Licensing Preferences.
- When design points are configured to be updated via RSM, the **Solution** cell cannot also be updated via RSM. In order to update design points via RSM, change the **Solution** cell update settings by setting the **Update Option** Solution Process property to **Run in Foreground**. For special circumstances, however, the update of both the **Solution** cell and design points via RSM can be enabled. For assistance with enabling this functionality and configuring your system to support, go to the Support page of the ANSYS Customer Portal and submit an online support request. For further information about tutorials and documentation on the ANSYS Customer Portal, go to https://support.ansys.com/docinfo.
- 1. First, configure RSM with the desired solve manager(s) and queue(s) to manage remote solutions. Once set up, these solve managers and queues can be used for all subsequent remote solutions. See the Remote Solve Manager User's Guide for detailed information on setting up RSM.
- 2. Right-click the **Parameter Set** bar and select **Properties** to view the **Design Point Update Process** settings. If the **Properties** view is already visible, click the **Parameter Set** bar and the **Properties** view will refresh with the design point settings.
- 3. In the **Properties** view, specify the **Design Point Update Process** settings. These settings will initially be populated based on your selections in **Tools> Options> Solution Process**. See Solution Process (p. 19) for more information. You can choose different settings here if the default settings are not appropriate. For design points to be updated via RSM, set **Update Option** to **Submit to Remote Solve Manager** (and then specify an available **Solve Manager** and **Queue**).
- 4. For the **Job Submission** property, select one of the following options:
 - One Job for All Design Points -- All design points are submitted as a single job to RSM.
 - One Job for Each Design Point -- Each design point is submitted as a separate job to RSM (simultaneous parallel updates).

• Specify Maximum Number of Jobs -- Design points are divided into groups and submitted in multiple jobs, up to the specified maximum number of jobs. (You can look at the RSM List view to determine which design points are assigned to each job.) If you select this option, the Maximum Number of Jobs property is enabled, allowing you to specify the maximum number of jobs that can be created.

Note

The **Maximum Number Jobs** property for the Compute Server (set on the **General** tab of the RSM **Compute Server Properties** dialog) specifies how many jobs can be run on the Compute Server at one time and takes precedence over the **Maximum Number of Jobs** property for the RSM Client machine. For example, if you set this property to 16 on the RSM Client but the maximum number of jobs is set to 10 on the Compute Server, only 10 jobs can run concurrently; the remaining 6 jobs will stay in the queue until resources become available and the next group of jobs can be submitted.

- 5. For the **Component Execution Mode** property, select one of the following options:
 - **Serial** All components participating in the update will run in serial mode.
 - **Parallel** All components participating in the update which support this setting will run in parallel mode.
 - If you select **Parallel**:
 - → The **Max Number of Processes per Job** property allows you to specify the maximum number of processes to be used in the solver for each job in the update. For jobs being sent to a cluster, the value entered for this property also determines how many cores are allocated on the cluster for each job.
 - → For jobs not being sent to a cluster, the **Max Number of Processes per Job** property has no effect on core allocation. The resource allocation is determined by the **Maximum Number of Jobs** property in the Compute Server used to submit the job. RSM considers each design point update job as a single job, although the component will use more resources.
 - At ANSYS 14.5, submitting a design point update to RSM is supported by the **Solution** (or **Analysis**) component update for the Mechanical APDL, Mechanical, Fluent, CFX and Polyflow solvers. These settings will override any parallel or serial settings defined at the component level. When updating a component, system or project, the **Parameter Set** properties for a Design Point update are ignored. The product specific settings that are overridden for a Design Point update via RSM are listed below.
 - → For Mechanical APDL:

The following properties on the Mechanical APDL **Analysis** cell are ignored for Design Point update via RSM:

- Processors
- Distributed
- MPI Type
- Machine list

→ For Fluent:

The following properties on the Fluent **Solution** cell are ignored for Design Point update via RSM:

- · Use Job Scheduler
- Run Parallel Version

(Visible under Parallel Run Settings when RPV=True) Number of Processes Use Shared Memory Machine Specification (visible when USM=false) Machine List (visible when USM=false)

Notable exceptions in the parallel run settings group are the following properties whose effects remain enabled if set in all situations.

Interconnect MPI Type

→ For CFX:

The following properties in the CFX-Solver Manager or CFX-Pre **Solution** cell are ignored for Design Point update via RSM:

· All settings under the Parallel Environment tab.

Run mode
All host information and partition weighting

→ For Polyflow:

The following properties in the Polyflow **Options** accessed via **Solution** cell preferences are ignored for Design Point update via RSM:

- Number of Processes
- → For Mechanical:

The following properties in the **Advanced** section of a Mechanical Solve Process Setting are ignored for Design Point update via RSM:

- Distributed Solution (if possible)
- Max number of utilized processors

Note

The **Component Execution Mode** and **Max Number of Processes per Job** settings are applied to all components participating in the update and cannot be overridden by component-level settings. To learn how to use component-level settings, please see the information in the step below.

- 6. For the **Exported Design Point** property, select one of the following options:
 - **Update parameters** -- Only parameters are updated for exported design points. Default value.
 - **Update full project** -- Full project is updated for exported design points.
- 7. At ANSYS 15.0, design point updates to RSM include new component override settings for individual systems using the Solution or (Analysis) component for Mechanical APDL, Mechanical, Fluent, CFX and Polyflow systems. These settings will override the Component Execution Mode and Max Number of Processes per Job defined in the parameter set. Before updating a component, right-click the Solution or (Analysis) component to expose the properties information. Under Restriction for Design Point Update Via RSM, enable Serial Execution Only if solving in serial. If solving in parallel, enable Specify Number of Processes Restriction and enter the number of processes in the text field below, across from Number of Processes Used Not to Exceed.

Note

These settings will only override the serial/parallel settings for that specific system.

 For Mechanical APDL and Mechanical, if solving in parallel, enable Specify Number of Processes Restriction, enter the number of processes in the text field below, across from Number of Processes Used Not to Exceed and enable Shared Memory Parallel.

Note

Shared Memory Parallel allows jobs to strictly run in parallel on the master node. It does not run jobs in distributed parallel.

8. You can specify **Geometry** in the **Pre-RSM Foreground Update** property if you want to update your geometry locally prior to submitting design point update(s) to RSM.

Note

Pre-RSM foreground update of geometry is necessary when design points will be updated simultaneously via RSM; the geometry update infrastructure does not support simultaneous design point updates in the same source geometry. If the project includes a **Geometry** component and you have set the **Default Job Submission** property either to **One Job per Design Point** or to **Specify Maximum Number of Jobs** (with **Maximum Number of Jobs** set to a value greater than 1), ANSYS Workbench will ignore the **Pre-RSM Fore-ground Update** property setting and the geometry will be updated locally in the foreground before your design points jobs are sent to RSM.

Pre-RSM foreground update of geometry may also be useful in other situations, such as:

- Your RSM Compute Servers are not configured to perform geometry updates (for example, if they do not have the necessary CAD software installed).
- You are submitting design point updates to a Compute Server that may not have the same licensing preferences as your local machine. For example, if you are submitting a geometry update that requires BladeModeler licensing to RSM, but the license preference of the remote Compute Server is set to DesignModeler, the geometry cannot be updated on the remote

machine. The remote licensing issue can be addressed by a local geometry-only update prior to the submission of the design point update.

- 9. Save the project. If you are working in an archived project, you will need to save the project to a permanent location.
- 10. Initiate the update of the desired design point(s) (see Updating Design Points (p. 103)). The project will be archived, submitted to RSM for remote solution, and the remote data will be retrieved periodically as the design point updates complete. Should a design point update fail, any errors will be reported to the **Messages** view.

When updating design points via RSM, each output parameter that was out of date when the design

point update was initiated will be shown in a pending state () in the **Table of Design Points**. ANSYS Workbench will periodically query RSM and will refresh any design point updates that have completed since the previous query. Design point updates that have not yet completed will continue to be shown in a pending state.

Design points that have been updated via RSM need to be reintegrated into the project as the updates complete. In order to do so safely and ensure the integrity of the data, ANSYS Workbench restricts or disables several GUI operations during pending RSM design point updates:

- · Open editors may automatically close.
- Drag and drop from the **Toolbox** is disabled.
- Most context menu, toolbar, and menu selections are disabled.
- Accessing Properties via a context menu is allowed, but properties in the Properties view cannot be modified.
- Input parameters cannot be modified.
- All File menu options except Exit and Save are disabled.

Related Topics:

Aborting or Interrupting an RSM Design Point Update Exiting a Project during an RSM Design Point Update Note for Using Windows LSF Scheduler Product-Specific Limitations

Aborting or Interrupting an RSM Design Point Update

You can abort or interrupt an RSM update in progress. In the **Progress** view, while the **Status** column

displays the message Waiting for background task, click on the interrupt icon () in the **Progress** column. You will be prompted to either abort or interrupt the update, or to cancel the dialog.

- If you select **Abort**, the RSM job is stopped (if it is already running) or removed from the queue (if the job is not yet running). No data will be returned to the project.
- If you select **Interrupt**, the RSM job is stopped (if it is already running) and any available data will be returned to the project. If the job has not yet started, interrupting behaves the same as stopping, and no

data will be returned to the project. A message with this information will be displayed in the **Messages** view.

• If you select **Cancel**, the dialog is dismissed and no action is taken.

Exiting a Project during an RSM Design Point Update

You can exit a project while an RSM design point update is in progress (i.e., one or more RSM jobs are running in the background and the **Progress** view displays a project **Status** of **Waiting for background task**).

In order for RSM jobs to continue to run after you exit the project, the project must be saved at least once after the design point update job was initiated.

If you attempt to exit a project while a design point update job is still running, the following scenarios will cause a dialog to display, allowing you to specify whether you want to save the project before exiting:

- You have never saved the project at any time after the same design point update job was initiated.
- You have saved the project at least once after the same design point update was initiated, but design point results have been retrieved since your last Save.

In either of these cases, if you do not save the project before exiting:

- All design point results retrieved since the last Save operation will be lost. If the project has been saved at least once since the update job was initiated, however, the results can be retrieved again when the project is reopened.
- If the project has never been saved after the update job was initiated, all RSM jobs will be aborted and will show a **Status** of **Cancelled** and the Cancelled icon () in the RSM **List** view; the asterisk on the icon indicates that the job has also been released. When you reopen the project, it will be in the state of your last manual save.
- If the project has been saved at least once after the update job was initiated, RSM jobs that are queued and running will continue to run after you exit. When you reopen the project, you can resume the update of the pending jobs to reconnect and download the results.

If you do save the project before exiting:

- Retrieved design point results will be saved to the project.
- RSM jobs that are queued and running will continue to run after you exit.
- Jobs for which the results have been saved will show a **Status** of **Finished** and the Finished icon (iii) in the RSM **List** view; the asterisk on the icon indicates that the job has also been released.
- Jobs for which results have not yet been saved are not released upon exit. When you reopen the project, you can resume update of the pending jobs to reconnect and download the results.

Note for Using Windows LSF Scheduler

If you are doing a design point update via RSM using Windows LSF scheduler, you may see lengthy delays with the remote updates finishing due to shared licensing. The shared licensing handler started by one design point update job can remain running after the completion of that job because it is also

being used by the next job, if that second job starts before the first job finishes. In this situation, both jobs are using the same shared licensing handler (started by the first job), and LSF does not recognize that the first job finished because the handler is still running.

In extreme circumstances, if multiple jobs are started with no break between them, then the first job that started the shared licensing handler cannot finish until the last job finishes and closes the shared licensing handler, potentially resulting in a long delay.

If you are using Windows LSF scheduler, we recommend that you either:

- · Make sure you run only one design point update job at a time on any execution node, or
- Turn license sharing off by setting the environment variable **ANSYSLI_FORCE_NOSHARE_UNDER_RSM**=1 on the local computer that initiates the update. Please be aware that setting this environment variable will cause the application to require one license per system in the project.

Product-Specific Limitations

Some ANSYS, Inc. products have additional limitations when submitting design points updates via RSM:

Rigid Body Dynamics and Explicit Dynamics

Rigid Body Dynamics and Explicit solvers always use RSM for update of the **Solution**, so it is not possible to update design points via RSM until special steps have been taken to enable update of **Solution** via RSM within design point update via RSM. For assistance with enabling this functionality and configuring your system to support, go to the Support page of the ANSYS Customer Portal and submit an online support request. For further information about tutorials and documentation on the ANSYS Customer Portal, go to http://support.ansys.com/docinfo.

Ansoft

Projects that include Ansoft systems will return updated parameter values but will not return the detailed solution for the 'Current' design point.

Third-Party CAD Systems

Projects that include geometry parameters that rely on third-party CAD systems will not update the geometry unless the CAD system is accessible on the compute server.

Mechanical

Design point updates via RSM can fail for a class of problems (typically involving a Structural system linked to a Modal or other type of Mechanical system) that meet the following criteria:

- A non-parameterized upstream Mechanical system provides solution data to a parameterized downstream Mechanical system and both systems share the same geometry/model.
- Updates are performed via pre-RSM local update.

The parameters for some of the design points will not be computed and will be marked as errors in the design point table.

Workarounds:

- Select the Enable Legacy Solve check box (Tools > Options > Mechanical), or
- In Parameter Set properties, set the Pre-RSM Foreground Update property to None and the Job Submission property to One Job for All Design Points.

Reserving Licenses for a Design Point Update

To ensure that you have sufficient licenses available for the duration of your design point study, you can reserve the licenses you will need for design point updates.

To reserve licenses for a design point update:

- 1. Set up your design point study as you normally would.
- 2. Right-click the **Parameter Set** bus bar and select **Properties**.
- 3. In the Properties view, select **Reserved** from the **License Checkout** drop-down.
- 4. Click **Select Licenses** from the **Reserved License Set** drop-down.
- 5. In the **Select Licenses** dialog, click on each license under the **Available Licenses** tab that you want to reserve and click **Add**. You can select multiple licenses at one time by holding down the **CTRL** key as you click each license. The licenses you selected appear in the **Reserved Licenses** panel. You can choose to filter the licenses shown by license type (Solver, PrepPost, Geometry, etc.)
 - When using an ANSYS HPC Parametric Pack license with either ANSYS HPC or ANSYS HPC Pack licenses, the number of licenses shown in the **Concurrent Licenses** column indicates the total number of HPC task available for the simultaneous design point update. For example, an HPC Parametric pack license used with eight HPC licenses will show 32 HPC tasks available because a single HPC Parametric Pack license enables four simultaneous design points, each of which can use up to eight HPC tasks. See Using HPC Parametric Pack Licenses (p. 116).
- 6. You may be able to see what licenses were used for this study in previous updates by clicking the **Used Licenses** tab. See Tracking Licenses (p. 115). You can add licenses to the reserve from this list as well.
- 7. To reserve more than one license of a given type, click on the license in the Reserved Licenses panel and either click Add multiple times or type in the number of licenses in the Change Number Selected box at the bottom of the panel.
- 8. When you have selected all of the licenses you need to reserve, click **OK**. The selected licenses will not be checked out at this time; license checkout occurs only when you begin the update.
- 9. Update your design points (either directly, or as a result of updating Design Exploration systems and components). The selected licenses will now be checked out and held for the duration of the update.

Restrictions

- You will see only those licenses that are available on license server machines that are in your license server path specification. You will not see or be able to reserve licenses from other license servers.
- All machines used in your design point study must use the same license server.
- If the license server you are using is part of a triad and the triad's master server goes down, subsequent license checkouts will behave as standard checkouts, rather than checking out licenses from the reserve pool.
- You do not need to reserve licenses for DesignXplorer components because DesignXplorer does not check licenses out of the reserve pool.

- If design points are being updated on remote resources using the ANSYS Remote Solve Manager (RSM), then the Compute Servers must be accessing the same license server as the source project.
- Do not use the **ANSWAIT** environment variable when reserving licenses.
- Reserved licensing can be disabled by your corporate license administrator. You will be warned when you try to use this feature if it has been disabled.

Special Cases

You might need to reserve a Geometry license even when the **Geometry** cell is not parameterized and is up-to-date, if either of the following situations is true:

- If the project contains CAD geometry or any other geometry that is not managed by the **Geometry** cell but rather by a downstream **Model** or **Mesh** cell, and the geometry is actually parameterized via the **Model** or **Mesh** cell.
- If an Engineering Data cell is parameterized and shares a model or mesh downstream with an unparameterized Geometry cell.

If you are using reserved Mechanical APDL licenses with RSM on Linux machines, AND you have the **ANSYS150_PRODUCT** environment variable set in a global login startup script (that is used by all users on a machine), you may see update failures caused by license checkout errors. You should remove the **ANSYS150_PRODUCT** environment variable from any global login startup scripts. Note that if the **ANSYS150_PRODUCT** environment variable is set in a user's local startup script, or is set manually at runtime or via the launcher, the reserved licenses will run correctly.

Related Topics:

Tracking Licenses
Returning Reserved Licenses
Using HPC Parametric Pack Licenses

Tracking Licenses

ANSYS Workbench now tracks licenses that are used during an update. You can see what licenses were used for any cell by viewing that cell's properties. The licenses used are shown in the **Last Update Used Licenses** field in the properties. You may find it useful to run the update of at least one design point as you normally would and then review the licenses that were actually used so that you know which licenses need to be reserved for a future study. License usage is tracked only for an Update operation in a component to which reserved licensing functionality may be applied; it is not tracked if a cell becomes up-to-date as the result of an Edit operation or for updates in DesignXplorer components.

Note

License usage is not tracked for DesignXplorer components; DesignXplorer uses reserved licenses indirectly by initiating design point updates.

License tracking may not return the information in some cases, such as if a cell's editor was open at the time of the update. In this case, you will see Not Applicable in the **Last Update Used Licenses** field,

even though a license was used. To ensure that tracking captures information for all components, you should close all cells' editors before updating.

Note

License tracking can be turned off by your corporate license administrator. If you run a study and do not see which licenses were used, please see your license administrator.

Returning Reserved Licenses

If you are running a design point study and are using reserved licenses, you may need to free licenses if one or more design point runs hang or do not complete successfully (for example, if you delete a project that contains pending updates using reserved licenses). This situation should rarely occur. Typically, licenses will be released if a design point fails. To free a reserved license, select **Tools> Release Reserved Licenses**. Select the projects for which you want to release licenses and click **Release Selected**.

This utility removes any free or hung licenses in the reserve; if any of the reserved licenses are still in use, those running jobs will continue to completion or until they attempt to check out further licenses. The design point study will fail if it attempts to update further design points, since the reserves have been removed.

Using HPC Parametric Pack Licenses

ANSYS, Inc. offers additional license options called HPC Parametric Packs for ANSYS Workbench jobs requiring multiple design point updates. HPC Parametric Packs enable you to simultaneously update multiple design points of a single design study while using only a single license of each required base license. HPC Parametric Pack licenses can be used for design point updates initiated from DesignXplorer. To use HPC Parametric Pack licenses, you must use the ANSYS Workbench reserved licensing feature.

HPC Parametric Packs enable a specific number of simultaneous design points, as follows:

- 1 HPC Parametric Pack: 4 simultaneous design points
- 2 HPC Parametric Packs: 8 simultaneous design points
- 3 HPC Parametric Packs: 16 simultaneous design points
- 4 HPC Parametric Packs: 32 simultaneous design points
- 5 HPC Parametric Packs: 64 simultaneous design points

You can use a maximum of five HPC Parametric Pack licenses per design study.

If you reserve more than one license of an individual licensed product, the HPC Parametric Pack license will multiply only one of the licenses (allowing four simultaneous design points when used with 1 HPC Parametric Pack). If you have multiple licenses specified in the Reserved Licenses panel when you add an HPC Parametric Pack license, the number of licenses will be decremented to one.

HPC Parametric Packs also work in conjunction with ANSYS HPC and ANSYS HPC Pack licenses. Use the HPC licenses to enable multiple parallel processes to be used for each design point.

For example, if you specify that you want to run the ANSYS Mechanical solver for the design point study using eight parallel processes for each design point, you must reserve the necessary HPC license(s) to enable eight parallel processes (one HPC Pack or six processes of ANSYS HPC). The HPC Parametric Pack

license allows all design points to run using eight-way parallel processing. If n design points are updated simultaneously, this scenario will use nx8 cores, while consuming only one HPC Pack or six processes of ANSYS HPC, as well as the license that enables the Mechanical solver.

If a design point update involves more than one solver, the parallel licenses that you reserve are available for use with both solvers running sequentially. For example, if you define a design point study that uses both an ANSYS Mechanical and an ANSYS Fluent license, and you specify that you want to use 8 parallel processes for Mechanical and 32 processes for Fluent, you must reserve sufficient HPC licenses to enable 32 parallel processes (2 HPC Packs or 32 processes of ANSYS HPC). The HPC Parametric Pack license then allows all design points to run using up to 32–way parallel with both solvers involved in the design point study. If you update n points simultaneously, this scenario will use nx8 cores for ANSYS Mechanical and nx32 cores for ANSYS Fluent, while consuming only 2 HPC Packs (or 32 ANSYS HPC processes).

See HPC Parametric Pack Licensing in the *Installation and Licensing Documentation* for additional details about using HPC Parametric Packs.

Design Point Update Data

When design points are updated, DesignXplorer provides the following methods of retaining parameter data for solved design points:

- It saves design point data to the design point cache. DesignXplorer reuses data from the cache when you preview or update a Design Exploration system. For more information, see Cache of Design Point Results in the *Design Exploration User's Guide*.
- It exports design point data into design point log files. You can import the log file back into DesignXplorer as needed. For more information, see Design Point Log Files in the *Design Exploration User's Guide*.
- It allows you to export all the design point values to an ASCII file. This functionality is available for
 DesignXplorer charts or tables containing design point data and for the **Table of Design Points**. The
 parameter values in the file can then be used by other programs for further processing or imported
 into DesignXplorer as needed. For more information, see Exporting Design Point Values to an ASCII
 File (p. 106).

Note

Both the design point log files and the ASCII file are formatted in DesignXplorer's "Extended CSV File Format." For details on the file formatting, see Extended CSV File Format in the DesignXplorer User's Guide.

Design Point States

On the **Parameters** tab, the output parameters that display in the **Outline** and **Table** views display states. Their states are indicated visually, as follows:

- Output parameters that are up-to-date have no icons; only the value of the output parameter is shown.
- Output parameters that are out-of-date are indicated by the **Update Required** icon (). Either they have not been solved or local data has changed, so an update is needed.

• Output parameters that have failed to update are indicated by the **Update Failed, Update Required**() icon.

During cell, system, project, and both local and remote design point updates, the states of output parameters are updated in the **Parameters** tab **Outline** and **Table** views to reflect the results of the update.

Changes elsewhere in the project could cause the state of some, but not all, output parameters to go out-of-date and require update. In those cases, only those output parameters affected by the changes will be shown as update required. Output parameters that are not affected by the changes will remain up-to-date.

Changes to derived parameters or associated expressions are recalculated immediately rather than requiring an update. If you add or change derived parameters, other parameters will not be affected by this change, and so will not require an update.

Design points that are being updated via RSM will appear in a **Pending** state in the **Table of Design Points** while the remote design point update is in progress. When each finished design point is retrieved, the associated output parameters will be updated to reflect the results of the update (if successful).

Working with ANSYS Workbench and EKM

The ANSYS Engineering Knowledge Manager (EKM) is a simulation process and data management (SPDM) software system that enables you to store, share, report, and operate on your simulation data in an accessible, web-based environment. While EKM can operate as a standalone system, its integration with ANSYS Workbench streamlines the process of storing, retrieving, and sharing your Workbench projects within an EKM repository. Through EKM, you can:

- Archive completed projects or store works in progress to local or remote storage
- Share and collaborate on your projects
- Change the permissions of projects or place them under versioning control
- Search projects based on names, dates, simulation type, or other criteria
- Retrieve your own projects or those shared by other users.

ANSYS EKM provides access to a simulation data repository which resides on a dedicated server for enterprise-level data management. For further information on installing EKM servers and creating repositories, refer to the ANSYS EKM Installation Guide. Within ANSYS Workbench, you can establish a connection to one or more EKM repositories to provide flexible management of local, remote, and/or shared data.

Once an ANSYS Workbench project has been stored to a repository, you can use the EKM web interface to perform the full set of EKM activities, including reporting, managing access permissions, or remote access of ANSYS Workbench functionality such as the execution of design point runs or performing optimization studies using ANSYS DesignXplorer. For a complete description of EKM capabilities, refer to the ANSYS EKM User's Guide.

This section covers the following topics:
Creating a Repository Connection
Launching EKM with a Web Browser
Working with Existing Repository Connections
Registering an ANSYS Workbench Session with an EKM Repository
Working with ANSYS Workbench Projects Saved in an EKM Repository
Importing Repository Files
Troubleshooting

Creating a Repository Connection

Before you can work with a project from the EKM repository, you must create a connection between a local ANSYS Workbench installation and an EKM repository.

A connection contains information for the server hosting the EKM repository to which you want to connect, the workspace within the repository, and the user credentials used to log onto the repository.

To create a connection from Workbench:

- 1. From ANSYS Workbench, open the **Save to Repository** dialog (File> Save to Repository) or **Open Project from Repository** dialog (File> Open Project from Repository).
- 2. If no connections exist, you will be prompted to create one. Click the **Create New Connection** button on the resulting dialog box. Alternatively, you can right-click inside the dialog and select **Create Connection**.
- 3. Follow the steps in the **Create New Connection** wizard.

For example, you may have one connection where users store their personal archives (a workspace with the server in Individual mode), and another connection where you store and access projects to collaborate with the rest of your workgroup (a workspace with the server in Shared mode).

Note

With an Individual workspace, you will only be able to see your own files, stored in your **My Data** directory. With a Shared workspace, you will be able to see both your own files and the shared files within the repository, stored in the **My Data** and **Repository** directories, respectively. For more information, see the ANSYS EKM User's Guide.

Once a connection is created, you can use the ANSYS Workbench **File** menu to interact with the EKM repository.

Launching EKM with a Web Browser

To simplify access to advanced EKM functionality, you can select **File** > **Launch EKM Web Client...** from ANSYS Workbench to open the EKM Web Client in your default browser. If you have more than one connection defined, you will first be prompted for which connection you want to open. From here, you can access any EKM functionality as described in the ANSYS EKM User's Guide.

Working with Existing Repository Connections

It is possible to create multiple connections to the EKM repository using the method described in Creating a Repository Connection (p. 119). Connections that you have created are retained and display in a connections list on various EKM dialogs accessed via the **File** menu (the **Save to Repository**, **Open Project from Repository**, and the **Register Session with Repository** dialogs). If there is an open connection, it shows a **Status** of **Opened** in the connection list.

Opening a Connection

If you want to open a connection:

- 1. Right-click on the connection you want to open.
- 2. Select **Open Connection** from the context menu.

The opened connection will show a **Status** of **Opened** in the connections list. Only once connection can be opened at one time, so the previously opened connection will be closed.

Closing a Connection

If you want to close a connection without opening another one:

- 1. Right-click on the open connection.
- 2. Select **Close Connection** from the context menu.

The **Opened** status is removed and the license is released, but the connection name is retained on the connections list.

Deleting a Connection

If you want to delete a connection:

- 1. Right-click on the connection you want to remove.
- 2. Select **Delete Connection** from the context menu.

The connection name is removed from the connections list. The connection must be recreated in order to use it again.

Registering an ANSYS Workbench Session with an EKM Repository

The EKM **My Jobs** folder provides tools that allow you to set up and execute jobs in EKM. Once a job is displayed in the **My Jobs** folder, you can monitor job status and view job details directly from the folder. Integration between the EKM Web Client and EKM Mobile application gives you additional options for monitoring, modifying, and managing jobs.

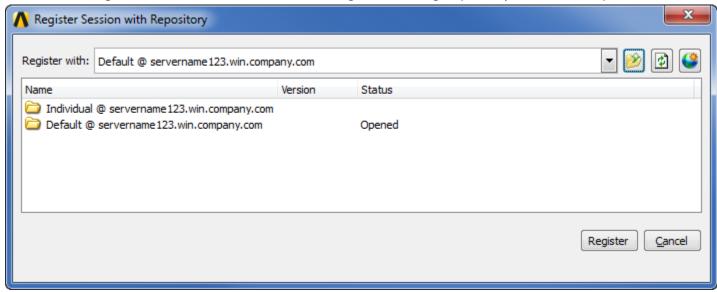
ANSYS Workbench allows you to register your current Workbench session with the EKM repository so that the session displays in the **My Jobs** folder, as well. Once the session has been successfully registered, a job is created for it in the EKM **My Jobs** folder. The job allows you to perform and monitor various Workbench operations remotely via the EKM Web Client or EKM Mobile application.

To register your current ANSYS Workbench session with an EKM Repository:

- 1. Save your Workbench project.
- 2. Select the **File > Register Session with Repository** menu option.
- 3. If you don't currently have an open repository connection, the **Open Connection** dialog displays. Create a new connection as described in Creating a Repository Connection (p. 119) or open an existing connection as described in Opening a Connection (p. 120).
- 4. Once a repository connection is open, the **Register Session with Repository** dialog displays. The **Register with** field contains the name of the currently selected repository.

Go up one directory to see other connections. In the example below, there are two connections: one to a Default workspace (with the server in Shared mode), and one to an Individual workspace (with the sever in Individual mode). In this example, you can see that the connection to the Default workspace is opened. For more information on workspace types and server modes, see the ANSYS EKM User's Guide.

You can register any of the available connections on the connections list. For more detailed information on working with the connections list, see Working with Existing Repository Connections (p. 120).



- 5. Select the connection you want to register and click the **Register** button.
- 6. If the registration was successful, the following dialog displays, indicating the name of the EKM server.



7. Click OK.

Now that the session has been registered, you can use the EKM Web Client or EKM Mobile application to view and work with the registered session. No other users will be able to access the job. Since the registration is associated with the Workbench session, rather than the project, the registration is valid for the duration of the session, as long as the session is active; you can open existing or create new projects within the open session without needing to repeat the registration. Once the session is closed, you must register it again upon reopening.

For information on working with registered sessions, see Working with Registered Workbench Projects in the ANSYS EKM User's Guide.

Working with ANSYS Workbench Projects Saved in an EKM Repository

When working with an EKM repository, you must understand the relationship between your local working version of the project and the copy stored in the repository. Even if a project has been stored to or opened from a repository, ANSYS Workbench always operates directly on the local working copy of the project.

ANSYS Workbench is aware of the relationship between the local project and the repository copy, and will help you manage your project and maintain consistency. When closing a changed local copy of a project you can update the repository copy. If you open a local project, and the copy in the repository is more recent, you have the option of downloading and updating to the repository version. This situation can happen if, for example, you are collaborating with another user on a project that is stored centrally in the repository, and that user has made recent changes to the project. Access control settings enable you to restrict project permissions or apply project versioning, helping to minimize the potential for conflicts or duplicate work.

ANSYS Workbench projects in an EKM repository are always stored as a Workbench Project Archive (with a .wbpz extension), which will provide a single consistent package that holds all project data. ANSYS Workbench manages the creation or extraction of projects from an archive for you. When saving to the repository, ANSYS Workbench will create a project archive and upload it to the repository. When opening a project from the repository, ANSYS Workbench will download the archive from the repository and extract it to the local directory specified in the Open Project from Repository dialog box.

The following sections discuss how to work with projects saved to an EKM repository. Each section assumes that you already have an open repository connection.

Saving a Project to an EKM Repository
Opening a Project from an EKM Repository
Sending Project Changes to the EKM Repository
Getting Project Changes from the EKM Repository
Managing EKM Repository Project Changes

Saving a Project to an EKM Repository

You can save your ANSYS Workbench project to an EKM repository for archive storage or to share with other users. Once the project has been saved to the repository, you can also perform additional EKM-based operations on it through the EKM Web Client.

To save a project to an EKM repository:

- 1. Create a new ANSYS Workbench project and set up the analysis to any desired status, or open an existing ANSYS Workbench project.
- Save the project, if necessary.
- 3. Select File > Save to Repository
- 4. Choose the repository location to which you want to save the project.

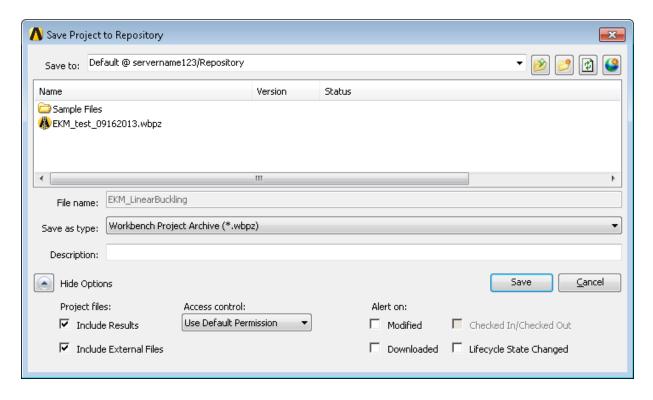
If the project already exists in the repository location, you cannot overwrite the existing project, so you will be given the option to save the local copy under a new filename. To save the newly named project to the repository, use the **File** > **Save to Repository** option.

- 5. Set the remote project filename and add a description if desired.
- 6. Click **Show Options** and select the **Include Results** or **Include External Files** options if desired.
- 7. Select one of the following options from the **Access control** drop-down:
 - Use Default Permission: Uses the default permissions for your account.
 - Get Exclusive Control: Gives you exclusive control of the project, preventing other users from editing
 it.

• **Place under Version Control**: Places the project under versioning control, allowing the file to be edited by only one user at a time. When you select this option, you can check out the project by selecting the **Check out** check box on the **Version Control** dialog box. Files under version control are identified by their version numbers in Workbench.

For more information on this option, see Version Control and Access Management in the *Engineering Knowledge Manager*.

8. Click the Save button.



The current project will be archived and uploaded to the EKM repository. Once the upload portion of the operation has started, you can continue with further analysis tasks or work with other projects. You will not be allowed to exit ANSYS Workbench until the upload has completed.

When you upload a file to the EKM repository, you will regain control of the ANSYS Workbench interface as soon as the upload completes; EKM will extract the metadata asynchronously. If the metadata extraction fails, you will not see any indication in ANSYS Workbench. However, the project object in EKM (viewed via the EKM Web Client) will indicate that metadata is missing.

Opening a Project from an EKM Repository

You can open a project from an EKM repository to load an archived project from an individual repository or access a project that has been placed previously on a shared repository.

To open a project from an EKM repository:

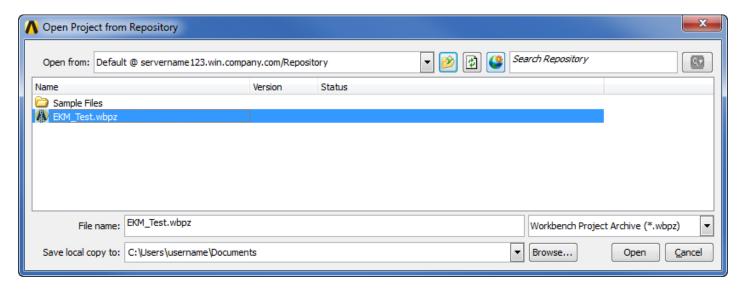
- 1. From ANSYS Workbench, select **File** > **Open from Repository**.
- 2. Browse to the repository location that contains the project and select it.
- 3. Change the location to which the local copy of the project is to be placed, if necessary.

- 4. Select **Open** to download the project from the repository.
- 5. If the project is under version control, check the project out when prompted.

Note

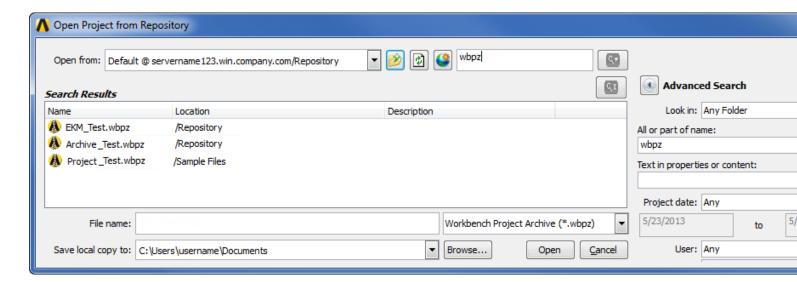
If the project is under version control, you must have the project checked out in order to send changes to the repository or modify the repository version of the project. You cannot check out a project that is already checked out by another user.

6. Extract the project to the specified location and open it in ANSYS Workbench.



The **Open Project from Repository** dialog box contains powerful search features to assist you in finding projects anywhere in the repository. To perform a basic search, enter text in the quick search field. The quick search will search the repository and show search results whose name, description, or additional metadata contains the specified text.

Select **Search Options** to perform an advanced search or to use a saved search from the EKM repository. From the advanced search panel, you can search on text or additional properties such as date, owner, or simulation types contained in the project. Use the **Saved Queries** option to perform predefined public or private queries that have been saved in the EKM repository. For additional information on saved queries, see Managing Queries and Search Results in the *Engineering Knowledge Manager*.

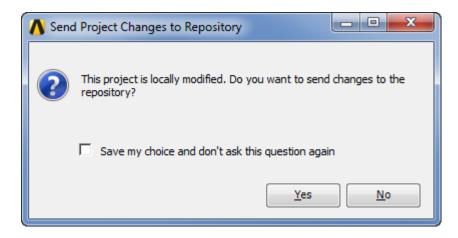


Sending Project Changes to the EKM Repository

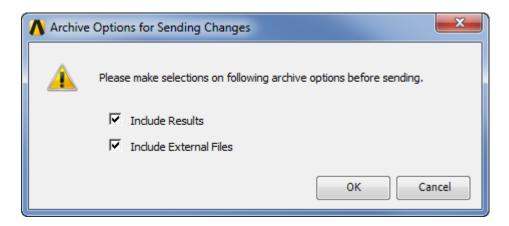
When working with a project that has been saved to or opened from an EKM repository, ANSYS Workbench enables you to update the repository with the current project version.

To update the repository with the changes made in a local project:

- 1. Save the project.
- Select the File > Send Changes to Repository menu item. You can use this option at any time to save your changes to the repository.
- 3. Alternatively, if you exit the project without sending your changes to the repository, the **Send Project Changes to Repository** dialog will ask if you would like to do so.
 - If you do not want to be prompted upon exit to send changes to the repository, select the Save my choice and don't ask this question again check box. Your preference will be saved to the Workbench Tools > Options dialog and will be used the next time you close a repository project without sending changes. For details on options for sending project changes, see Repository (p. 20).
 - Click Yes.



4. The **Archive Options for Sending Changes** dialog allows you to specify which changes to include in the send. You can select **Include Results** and/or **Include External Files**.



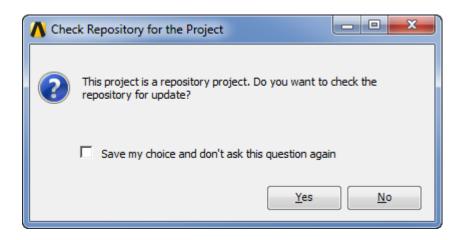
You cannot send changes to the repository if the project is checked out to another user, locked by another user, or you do not have write permissions to the project. When sending changes to the EKM repository for a project under version control, you can either keep the project checked out, check the project back in, or check the project in and then check it back out. When you check the project in, you have the option of adding comments. The project version will be updated upon check in.

Getting Project Changes from the EKM Repository

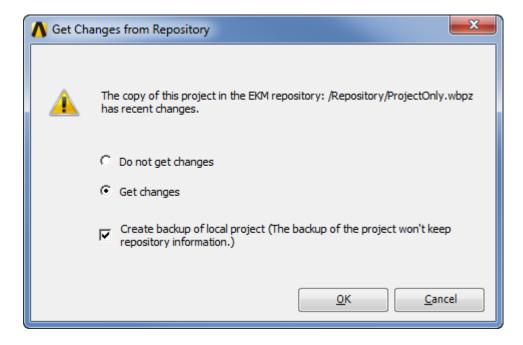
When you open an ANSYS Workbench project that has been previously opened from or saved to an EKM repository, ANSYS Workbench checks to determine if a more recent copy of the project exists in the repository. If a more recent copy exists, you are prompted and given the option to update your local copy with the version of the project in the repository. If the project is under version control and is not already checked out (either by another user or by yourself), you are given the option of checking the project out.

To update a project with the changes from the EKM repository:

- 1. Open an ANSYS Workbench project that has a more recent copy in the repository.
- 2. The **Check Repository for the Project Dialog** asks if you want check the repository for changes to the project.
 - If you do not want to be prompted to check for project updates in the future, select the Save my choice and don't ask this question again check box. Your preference will be saved to the Workbench Tools > Options dialog and will be used the next time you open a repository project. If you do not select this check box, ANSYS Workbench will check for project updates according to the preference set in the Options dialog. For details on configuring how Workbench checks for project updates, see Repository (p. 20).
 - · Click Yes.



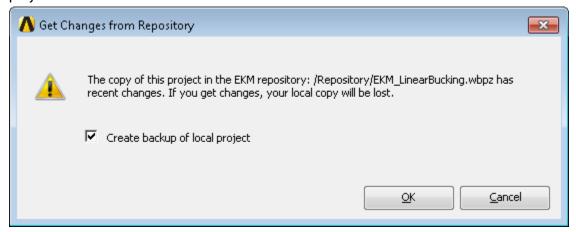
- If there are changes to the repository version of the project, the Get Changes from Repository dialog will ask if you want to download the changes to your local project.
 - Specify whether or not you want to download the changes.
 - Select the **Create backup of local project** check box if you want to save a copy of the project before downloading the changes. (Note that the backup project will not be under repository control.)
 - · Click OK.



For a project that is already open, you can also manually retrieve changes from the EKM repository at any time: To do so:

- In ANSYS Workbench, select File > Manage Repository Project > Refresh Control Status. This synchronizes the status of your local project with the status of the repository version.
- 2. Select **File > Get Changes from Repository** in ANSYS Workbench.
- 3. If there are changes to the repository version of the project, the **Get Changes from Repository** dialog displays and lets you know that getting changes will overwrite your local copy.

4. By default, the **Create backup of local project** check box is selected. If you leave it selected, a backup copy called projectname>_backup.wbpj will be created in the same local directory as the original project.



Managing EKM Repository Project Changes

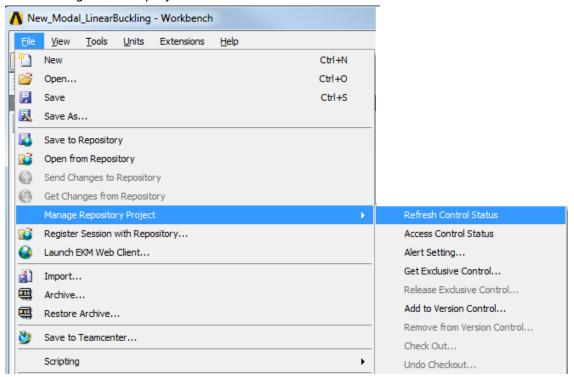
When you have a project opened from the repository, the **File > Manage Repository Project** menu includes the following options. Note that certain options may be disabled according the project status and your permissions.

- Refresh Control Status: Synchronize local project status with repository project status. Other Manage
 Repository Project menu options will then be become enabled or remain disabled according to the project
 status and your permissions.
- Access Control Status: View the current control status of the project. Selecting this option performs a refresh of the menu so that it shows the current status of the project in terms of exclusive control, version control, and checkout availability.
- Alert Setting: Specify alert settings for the project. Selecting this option launches the Alert Settings dialog, which enables you to specify that you will be notified by email when certain events occur (i.e., when the project is modified, downloaded, checked in/checked out, or when its lifecycle state is changed).
- **Get Exclusive Control**: Available concurrently with the **Add to Version Control** option. Selecting this option allows you to gain exclusive control of the project.
- Release Exclusive Control: Available only when you have exclusive control of the project. Selecting this
 option releases the exclusive control.
- Add to Version Control/Remove from Version Control: Can be available concurrently with the Get Exclusive Control option. When adding the project to version control, you have the option of checking out the project (which is necessary to send changes to the repository).
- · Check Out/Undo Checkout

To gain or release control over a project via the **File > Manage Repository Project** menu:

- 1. Load the project into ANSYS Workbench from the repository.
- 2. Select **File > Manage Repository Project > Refresh Control Status** to synchronize your local project settings with the project settings in the repository version of the project. Selecting this option refreshes the Workbench view of the repository, which may change the status of the exclusive control, version

control, and check out settings, depending on whether there are other users accessing the repository and working with the project files.



- 3. Select **File > Manage Repository Project > Access Control Status** to view the full status of the file in the **Access Control Status** dialog.
- 4. Select **File > Manage Repository Project > <action>**, where **<action>** is one of the control status-related options available to you. For example, you can:
 - Select **Add to Version Contro** to add the project to version control. If a project is already under version control, you can select **Remove from Version Control** to remove it.
 - Select **Get Exclusive Control** to gain exclusive control of the project. If the project is already under exclusive control, you can select **Remove from Exclusive Control** to allow others to access the project.

If the desired option is not available, you can view the repository to see which user has made the project file unavailable to you.

Note

When you make the desired change to the repository, you need to manually refresh your view of the repository to confirm the change of state.

Importing Repository Files

On the Workbench Project Schematic, certain system cells allow you to browse the EKM repository for files, which you can then import into a Workbench project. The **Browse from Repository** menu option may be available for any cell that already has the local **Browse** option.

Note

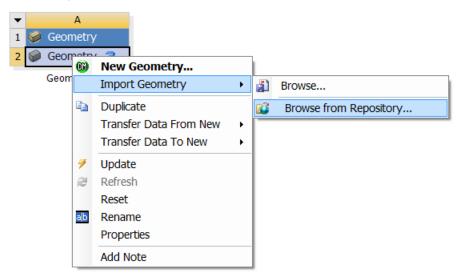
To import files, you must have a single open repository connection; multiple repository connections are not supported. For more information on opening a connection, see Creating a Repository Connection (p. 119).

In this example, we'll use the **Geometry** cell of a standalone **Geometry** system to import a data file into the project.

Importing the Repository File to a Project

To import an EKM repository file into your Workbench project::

1. In the Project Schematic, right-click the **Geometry** cell and select **Import Geometry > Browse from Repository**.



2. In the **Open from Repository** dialog, select the desired file and click the **Open** button.

Verifying the Repository File Import

You can verify that the file was successfully imported to your project by checking the project **Files** view. The repository file can be distinguished from local files by the EKM icon and the repository path in the **Location** column.

Although the **Files** view shows only the repository location, a copy of the imported file is saved and stored locally so you can continue working on the project without having a connection to the repository.

Checking for Newer Versions of Imported Files

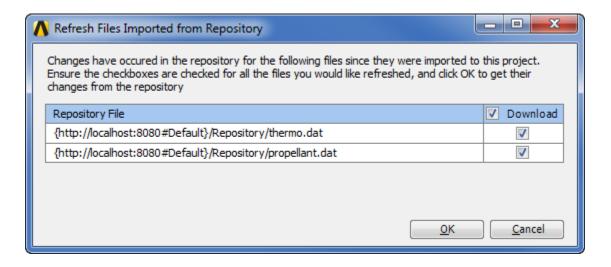
ANSYS Workbench allows you to check for newer versions of a file imported from the repository. The checks can be done either automatically (when the project is opened via the **File > Open** menu item) or manually (at any time via the **File > Get Changes from Repository** menu item).

You can also check for newer file versions manually at any time, as follows:

- 1. Select the **File > Get Changes from Repository** menu item.
- 2. The **Check Repository for Imported Files** dialog opens. Click **Yes** to check for newer file versions.



- 3. If you do not want to be prompted to check for updates to imported files, select the Save my choice and don't ask this question again check box. to save your preference to the Workbench Tools > Options dialog. Your preference will be saved to the Workbench Tools > Options dialog and will be used the next time you open a project contains files imported from a repository. For details on configuring how Workbench checks for updates to imported files, see Repository (p. 20).
- 4. The **Refresh Files Imported from Repository** dialog opens, showing a list of imported files with changes in the repository. To download the newer version of a file, select the **Download** check box for that file and click **OK**.
- 5. When the new file version is downloaded to the project, the state of an up-to-date cell changes to **Refresh Required**.



Note

While it is possible to download newer versions of imported files from the repository, you cannot send local changes to the files back to the repository. To change a file in the repository, open the EKM Web Client and overwrite the existing file with a newer version.

Troubleshooting

If you encounter problems connecting to or using an EKM Repository, review the following hints and tips. For additional help, also refer to Appendix N: Troubleshooting in the *Engineering Knowledge Manager*.

To be able to connect to an EKM repository, the EKM server must be running on the target machine.

Connect to Server Using Web Browser

If you encounter problems with connecting to an EKM server from within ANSYS Workbench, try opening a connection manually to the server using a web browser:

- 1. Open a web browser.
- 2. In the address bar, enter the full address to the EKM server. Assuming the server is running on the default port of 8080, the full address will be:

```
http://<server_name>:8080/ekm
```

3. If successful, the EKM Web Client for that server will be launched and you will be prompted for your login credentials.

See Launching the EKM Web Client in the Engineering Knowledge Manager for more information.

Disable your Pop-up Blocker in Web Browser

If you cannot connect with a browser, ensure the pop-up blocker is disabled or allows pop-ups from the EKM server.

Release 15.0 - © SAS IP, Inc. Al	Trights reserved - Conta	inc proprietary and	confidential inf	ormation
nelease 13.0 - @ 3A3 IF, IIIC. AI	i ngnis reserveu Contu	iiris proprietary ana	Commuentiarini	omination
-11	NSYS. Inc. and its subsid	l:		
OI A	an sy s. inc. ana its subsia	iaries ana attiliates.		

ANSYS Workbench Systems

The systems available in the **Project** tab **Toolbox** are divided into the following categories:

- Analysis Systems (p. 135) -- Complete systems with all the necessary component cells already defined and ready to be populated. For example, a Static Structural analysis system includes all the cells needed for the analysis, **Engineering Data** through **Results**.
- Component Systems (p. 149) -- Component building blocks which represent only a subset of a complete analysis. For example, you can use a Geometry component system to define your geometry and then connect the component system to several downstream systems, so component system can then be connected to several downstream systems, so that the downstream systems share the same geometry source. The Component Systems category also includes applications that open outside of ANSYS Workbench (rather than as a tab), allowing you to use Workbench to manage your analysis data and files. This can be useful for products such as Mechanical APDL, which uses numerous files during an analysis.
- Custom Systems (p. 240) -- Predefined templates for custom coupled systems, comprised of multiple analysis systems with predefined data connections. You can also create templates for your own custom system templates, which will then be stored and displayed as part of this category.
- Design Exploration (p. 242) -- DesignXplorer systems that can be added beneath the **Parameter Set** bar, allowing you to perform various design exploration studies.
- External Connection Systems (p. 243) -- enables you to integrate custom, lightweight, external applications and processes into the ANSYS Workbench Project Schematic workflow. Features exposed by the External Connection also allow you to perform automation and customization activities.

With the External Connection, you can integrate custom, lightweight, external applications; define User Interface (UI) elements, such as buttons in the Workbench Toolbar or entries in custom menus, and create the scripts that enable them; and create new systems to facilitate interaction with the Workbench Project Schematic.

Analysis Systems

One way to start an analysis in ANSYS Workbench is to select an analysis system from the **Toolbox**. When you select one of these analysis types, the corresponding system will appear in the **Project Schematic**, with all the necessary components of that type of analysis. Some analysis types offer different solvers, noted in parentheses. The features available can differ from one solver to another.

Available analysis systems include:

Design Assessment
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Fluid Flow (Polyflow)
Harmonic Response
Hydrodynamic Diffraction

Hydrodynamic Time Response

IC Engine

Linear Buckling and Linear Buckling (Samcef)

Magnetostatic

Modal and Modal (Samcef)

Random Vibration

Response Spectrum

Rigid Dynamics

Static Structural and Static Structural (Samcef)

Steady-State Thermal and Steady-State Thermal (Samcef)

Thermal-Electric

Throughflow

Transient Structural and Transient Structural (Samcef)

Transient Thermal and Transient Thermal (Samcef)

When you either double-click or drag an analysis system onto the **Project Schematic** it appears in the **Project Schematic** as a system. Components for that analysis type's system are listed as individual cells. For example, a typical structural analysis might have the following components/cells:

- Engineering Data (p. 252)
- Geometry (p. 253)
- Model/Mesh (p. 253)
- Setup (p. 254)
- Solution (p. 254)
- Results (p. 255)

Right-click on each cell to see a menu of actions that are available for that cell. Selecting an action may launch a separate application, if appropriate. When you've completed the necessary actions in that application, you can solve in the application or return to the **Project Schematic** to Update the project and continue. Updating a project allows other systems or other cells within the same system to acquire the newest information.

ANSYS Workbench provides templates for some of the commonly used coupled analyses, such as one-way FSI analyses, pre-stress modal, thermal stress, random vibration, and response spectrum. Select these templates from the Custom Systems area of the **Toolbox**.

You can also import databases from previous releases. See Importing Legacy Databases (p. 78) for instructions and restrictions on importing legacy databases.

Design Assessment

The Design Assessment analysis system provides the capability of performing a solution combination for static and transient structural analyses, and then performing post processing through a customizable script using additional geometry-associated data and extraction of custom results.

You will configure your Design Assessment Analysis in the Mechanical application, which uses the appropriate solver to compute the solution.

- Add static and/or transient structural analysis templates to the **Project Schematic** and attach the geometry.
- Add a Design Assessment analysis template by dragging the template from the Toolbox onto the last structural template in the Project Schematic, sharing the Engineering Data, Geometry, and Model cells.
- 3. The assessment type will then need to be set. This can be done in two ways, explained below.
- 4. Specify the Mechanical settings by right-clicking on the **Model** cell and choosing **Edit**.
- 5. In the Mechanical application window, complete your analysis using the application's tools and features.
- 6. Start the solution by selecting **Solve** from the Mechanical application or **Update** from the **Solution** cell in the **Project Schematic**.

Available Assessment Types

There are three supplied assessment types.

- FATJACK
- BEAMST
- · Solution Combination Only

You can also choose to define your own type by creating an attribute file for use with the system.

Note

- The default assessment type is Solution Combination Only.
- FATJACK and BEAMST options are only available if ASAS is installed.

For more information on creating the attribute file see The Design Assessment XML Definition File in the ANSYS Mechanical User's Guide.

How to Set the Assessment Type

Use of one of the two methods described below to set the assessment type for the system.

· Setup Cell Right Mouse Button Menu

Right-click on the **Setup** cell of the system and select **Assessment Type** from the menu. Here you can select either one of the pre-defined types or import a user defined XML file.

If you select to import a user defined type you will be presented the option to import the file by either browsing to it or by selecting one that has been browsed to previously from the list (if available).

To check which assessment type has been selected, there is a check box next to the pre-defined types on the menu that will display a check mark when they have been selected. If no check mark is visible then a user defined type has been selected.

Setup Cell Properties Panel

Select **View** then **Properties** from the main menu. This will display the **Properties** view in the workspace.

Now click on the **Setup** cell of the design assessment system and the **Properties** view will be updated to show the available options for the cell.

From here you can change the assessment type from the drop-down list in the **Design Assessment Settings** section. You can choose between the pre-defined types or to use a user-defined type. If you select the user defined option you will be presented with an open file dialog so you can choose the XML file you want to use. The name of this file will then be displayed in the **Properties** view.

Electric

An electric analysis supports Steady-State Electric Conduction. Primarily, this analysis type is used to determine the electric potential in a conducting body created by the external application of voltage or current loads. From the solution, other results items are computed such as conduction currents, electric field, and joule heating.

You will configure your electric analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

- 1. Add an electric analysis template by dragging the template from the **Toolbox** into the **Project Schematic** or by double-clicking the template in the **Toolbox**.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry.
- 3. Right click the **Setup** cell and select **Edit**, or double-click the **Setup** cell. This step will launch the Mechanical application.
- 4. In the Mechanical application window, complete your electric analysis using the Mechanical application's tools and features. See Electric Analysis in the Mechanical application help for more information on conducting an electric analysis in the Mechanical application.

Explicit Dynamics

- Add an explicit dynamics analysis template by dragging the template from the **Toolbox** into the **Project** Schematic or by double-clicking the template in the **Toolbox**.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry.
- View the geometry by right-clicking on the Model cell and choosing Edit, or double-clicking the Model
 cell. Alternatively, you can right click the Setup cell and select Edit. This step will launch the Mechanical
 application.
- 4. In the Mechanical application window, complete your explicit dynamics analysis using the Mechanical application's tools and features. See Explicit Dynamics Analysis in the Mechanical application help for more information on conducting an explicit dynamics analysis in the Mechanical application.

Fluid Flow (CFX)

ANSYS CFX enables you to perform fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You import the geometry and meshes, specify the materials, boundary conditions, and solution parameters, solve the calculations, view the results, then create reports using built-in tools.

To use ANSYS CFX as an analysis system in ANSYS Workbench:

- 1. Add a Fluid Flow (ANSYS CFX) analysis system by dragging the system from the Toolbox into the Project Schematic or by double-clicking the system in the Toolbox.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry. Alternatively, you can create the geometry in DesignModeler by right-clicking on the Geometry cell and choosing New Geometry.
- 3. Create a mesh by right-clicking on the **Mesh** cell and choosing **Edit**.
- 4. Specify the ANSYS CFX physics definitions in CFX-Pre by right-clicking on the **Setup** cell and choosing **Edit**. Alternatively, you can import a previously saved case file by right-clicking on the **Setup** cell and choosing **Import Case** > **Browse**.
- Right-click on the Solution cell and choose Update to start the solver. Alternatively, right-click on the Solution cell and select Edit, set the solver execution controls in CFX-Solver Manager, and start the solver.
 - You can also import an existing CFX-Solver Results file by right-clicking on the **Solution** cell and choosing **Import Solution** > **Browse**.
- 6. Analyze the results of the calculations in CFD-Post by right-clicking on the **Results** cell and choosing **Edit**.

For detailed information on working with ANSYS CFX, see ANSYS CFX in ANSYS Workbench in the *CFX Introduction*.

The Fluid Flow (ANSYS CFX) analysis system is also used as part of the FSI: Fluid Flow (ANSYS CFX) > Static Structural custom system.

Note

If you import a CFX-Solver Input File into a CFX **Setup** cell and:

- the reload file format is not a CFX-Solver Input File (for example, a DEF file)
- the Reload Options were not defined or did not contain a replacetype option

...you may need to modify the definition of the Mesh Reload Options as follows.

Example 1: Modifying the Mesh Reload Options

If a definition file contains a mesh from a GTM file and is imported into a **Setup** cell, the original mesh type is automatically changed to "CFX-SolverInput file". If the remeshing process is not explicitly told to import the GTM file as a GTM file, the remeshing process assumes that the mesh is also a "CFX-Solver Input file", rather than a GTM file.

This fails when the remeshing process runs. This is a limitation and needs to be modified by hand. To do this, add a line similar to the following to the DEF file:

Mesh Reload Options = "replacetype=GTM, notransform=false"

See Remeshing Tab in the CFX-Pre User's Guide for details.

Fluid Flow (Fluent)

Fluent allows for fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You specify the computational models, materials, boundary conditions, and solution parameters in Fluent, where the calculations are solved.

Use a Fluent fluid flow analysis system to apply a computational mesh to a geometry within Workbench, then use Fluent to define pertinent mathematical models (e.g., low-speed, high-speed, laminar, turbulent, etc.), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Fluent solves the mathematical equations, and the results of the simulation can be displayed in Fluent or in CFD-Post for further analysis (e.g. contours, vectors, etc.).

- 1. Add a fluid flow analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**. Alternatively, you can create the geometry in DesignModeler by right-clicking on the **Geometry** cell and choosing **New Geometry**.
- 3. Create a mesh by right-clicking on the **Mesh** cell and choosing **Edit**.
- 4. Specify the Fluent settings by right-clicking on the **Setup** cell and choosing **Edit**. Alternatively, you can import a previously saved Fluent case file by right-clicking on the **Setup** cell and choosing **Import FLUENT Case**.
- 5. Analyze the results of the calculations in CFD-Post by right-clicking on the **Results** cell and choosing **Edit**.

For detailed information on working with Fluent, see the Fluent User's Guide as well as the other online documentation available under the Help menu within Fluent. In addition, please see the Fluent in Workbench User's Guide.

Fluid Flow (Polyflow)

Polyflow allows for the analysis of fluid flows with free surfaces, complex rheology (including non-Newtonian behavior with viscoelasticity), heat transfer, and chemical reactions. The usage of Polyflow involves: specifying the computation models inside Polydata, which is the module for problem setup; and running the calculations using the Polyflow solver. Polyflow also comes with several useful utilities, which can be accessed via the right-click menu on the **Setup** and **Solution** cells. For example, Polymat can be used to calculate material properties, including viscoelastic parameters based on experimental data. For more details, please see the product documentation.

There are three Polyflow fluid flow analysis systems available in Workbench:

- The **Fluid Flow (Polyflow)** system provides the full simulation capabilities of Polyflow.
- The **Fluid Flow Blow Molding (Polyflow)** system provides only the application-specific capabilities of Polyflow that are suited to blow molding simulations.
- The **Fluid Flow Extrusion (Polyflow)** system provides only the application-specific capabilities of Polyflow that are suited to extrusion simulations.

Use a Polyflow, Blow Molding (Polyflow), or Extrusion (Polyflow) fluid flow analysis system to apply a computational mesh to a geometry within Workbench, then use Polydata to define pertinent mathem-

atical models (e.g., Generalized Newtonian, Viscoelastic, etc.), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Polyflow solves the mathematical equations, and the results of the simulation can be displayed in CFD-Post for further analysis (e.g. contours, vectors, etc.).

- 1. Add a Polyflow, Blow Molding (Polyflow), or Extrusion (Polyflow) fluid flow analysis system by dragging the system from the Toolbox into the Project Schematic or by double-clicking the system in the Toolbox.
- 2. Load the geometry by right-clicking the **Geometry** cell and clicking **Import Geometry** in the context menu that opens. Alternatively, you can create the geometry in DesignModeler by right-clicking the **Geometry** cell and clicking **New Geometry** in the context menu that opens.
- 3. Create a mesh by right-clicking the **Mesh** cell and clicking **Edit** in the context menu that opens. Alternatively, you can import a previously saved mesh by right-clicking the **Setup** cell and clicking **Import Mesh** in the context menu that opens. You can merge, scale, translate, and rotate the mesh by right-clicking the **Setup** cell and clicking **Polyfuse** in the context menu that opens.
- 4. You can define your preferences for Polydata by right-clicking the **Setup** cell and clicking **Preferences** and **Polydata** in the context menu that opens.
- 5. Specify the simulation setup by right-clicking the **Setup** cell and clicking **Edit** in the context menu that opens. Alternatively, you can import a previously saved Polyflow data file by right-clicking the **Setup** cell and clicking **Import Polyflow Dat...** in the context menu that opens. You have the option of specifying material data by right-clicking the **Setup** cell and clicking **Polymat** in the context menu that opens.
- 6. You can define your preferences for Polyflow by right-clicking the **Solution** cell and clicking **Preferences** and **Polyflow** in the context menu that opens.
- 7. Run the Polyflow calculation by right-clicking the **Solution** cell and clicking **Update** in the context menu that opens. You can check the status of the solver during or after the calculation by right-clicking the **Solution** cell and clicking **Polydiag** in the context menu that opens. You can open the listing file to see what Polyflow has done during or after the calculation by right-clicking the **Solution** cell and clicking **Listing Viewer** in the context menu that opens.
- 8. Analyze the results of the calculations in CFD-Post by right-clicking the **Results** cell and clicking **Edit** in the context menu that opens. You can generate plots of the solution data by right-clicking the **Solution** cell and clicking **Polycurve** in the context menu that opens. You can statistically postprocess the results of the solution data by right-clicking the **Solution** cell and clicking **Polystat** in the context menu that opens.

For detailed information on working with Polyflow, see the online help in Polyflow, as well as the separate ANSYS Polyflow User's Guide. In addition, please see the separate ANSYS Polyflow section in the ANSYS Workbench User's Guide.

Harmonic Response

In a structural system, any sustained cyclic load will produce a sustained cyclic (harmonic) response. Harmonic analysis results are used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations. This analysis technique calculates only the steady-state, forced vibrations of a structure, typically at a number of discrete points within a range of frequencies. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis.

You will configure your harmonic response analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

- 1. Add a harmonic response analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the Model cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.
- 4. In the Mechanical application window, complete your harmonic response analysis using the Mechanical application's tools and features. A mode-superposition harmonic analysis will automatically run the modal portion of the solution and cannot transfer data from a separate modal system in the Project Schematic. See Harmonic Response Analysis in the Mechanical application help for more information on conducting a harmonic response analysis in the Mechanical application.

Hydrodynamic Diffraction

Aqwa allows for the calculation of Wave Forces and Structure Motions in regular or irregular waves. You specify the geometry in DesignModeler, and Aqwa specific solution parameters within the Aqwa application where the calculations are solved.

Use an Aqwa Hydrodynamic Diffraction analysis system to apply a computational mesh to a geometry within the Aqwa application and produce a solution.

- 1. Add a Hydrodynamic Diffraction analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry. Alternatively, you can create the geometry in DesignModeler by right-clicking on the Geometry cell and choosing New Geometry.
- 3. Specify the Aqwa settings by right-clicking on the **Setup** cell and choosing **Edit**.
- 4. In the Aqwa application window, complete your Aqwa analysis using the application's tools and features.
- 5. Start the solution by selecting **Solve** from the Aqwa application or **Update** from the **Solution** cell in the Project Schematic.

You also have the option of importing a previously saved Aqwa .aqdb file by selecting **File > Import** from the Workbench toolbar, choosing files of Type **AQWAWB Database (*.aqdb)**, and navigating to the database file. See Importing Legacy Databases (p. 78) for additional information.

For detailed information on working with Aqwa in ANSYS Workbench, see Aqwa Introduction: What is Aqwa? available in the ANSYS online help. In addition, please see the separate Aqwa documentation that can be accessed from **Start > All Programs > ANSYS 15.0 > Help > AQWA > AQWA Reference 15.0**.

Hydrodynamic Time Response

Aqwa allows for the calculation of Wave Forces and Structure Motions in regular or irregular waves. You specify the geometry in DesignModeler, and Aqwa specific solution parameters within the Aqwa application where the calculations are solved.

Use an Aqwa Hydrodynamic Time Response analysis system to apply ocean environment forces (wind, wave, current) to a structure.

- 1. Add a Hydrodynamic Diffraction analysis template to the Project Schematic and attach a geometry to it.
- Add a Hydrodynamic Time Response analysis template by dragging the template from the Toolbox into the Hydrodynamic Diffraction system in the Project Schematic or by double-clicking the template in the Toolbox, sharing the **Geometry**, **Model**, and **Solution** cells from the Hydrodynamic Diffraction system.
- 3. Specify the Aqwa settings by right-clicking on the **Setup** cell and choosing **Edit**.
- 4. In the Agwa application window, complete your Agwa analysis using the application's tools and features.
- 5. Start the solution by selecting **Solve** from the Aqwa application or **Update** from the **Solution** cell in the Project Schematic.

You also have the option of importing a previously saved Aqwa .aqdb file by selecting **File > Import** from the Workbench toolbar, choosing Files of Type **AQWAWB Database (*.aqdb)**, and navigating to the database file. See Importing Legacy Databases (p. 78) for addition information.

For detailed information on working with Aqwa in ANSYS Workbench, see Aqwa Introduction: What is Aqwa? available in the ANSYS online help. In addition, please see the separate Aqwa documentation that can be accessed from **Start > All Programs > ANSYS 15.0 > Help > AQWA > AQWA Reference 15.0**.

IC Engine

IC Engine (Internal Combustion Engines in Workbench) is a customized application to setup and solve the flow inside an IC engine. IC Engine system is used for quantification of flow rate, swirl and tumble, and other flow parameters inside the engine during the engine cycle with moving geometry. IC Engine system uses ANSYS Fluent solver for fluid flow analysis.

- 1. Add an IC Engine analysis system by dragging the system from the Toolbox into the Project Schematic or by double-clicking the system in the Toolbox.
- 2. Select type of simulation and specify engine parameters in the ICE cell **Properties** and update the cell.
- Double-click the Geometry cell to open the DesignModeler. Load the geometry and click Input Manager
 to enter the geometry inputs required and generate the features. Then decompose the geometry by
 clicking Decompose.
- 4. Open the Meshing™ application by double-clicking, or selecting Edit from the context menu of the Mesh cell of IC Engine System. Once the geometry is loaded into the meshing application, click IC Setup Mesh and set the meshing parameters followed by IC Generate Mesh to create the mesh. Update the Mesh cell in the IC Engine System.

- 5. Open the ANSYS Fluent application by double-clicking or editing the **Setup** cell and enter the number of time steps in Fluent settings and run the case.
- 6. Analyze the results of the calculations in CFD-Post by double-clicking the **Results** cell.

For detailed information on working with IC Engine, see Internal Combustion Engines in Workbench.

Linear Buckling and Linear Buckling (Samcef)

Linear buckling (also called as Eigenvalue buckling) analysis predicts the theoretical buckling strength of an ideal elastic structure. This method corresponds to the textbook approach to elastic buckling analysis: for instance, an eigenvalue buckling analysis of a column will match the classical Euler solution. However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. Thus, linear buckling analysis often yields quick but non-conservative results.

You will configure your linear buckling analysis in the Mechanical application, which uses the ANSYS or the Samcef solver to compute the solution.

A linear buckling analysis must follow a prestressed static structural analysis. Follow the instructions in Static Structural and Static Structural (Samcef) (p. 147) to build a prestressed Static Structural system, and then follow the instructions below to build and link a linear buckling system.

- 1. From the Static Structural system, right click on the **Solution** cell and select **Transfer Data to New>Linear Buckling** .
- 2. A new linear buckling system is created, with the **Engineering Data**, **Geometry**, **Model**, and **Setup** cells linked from the static structural system.
- 3. Right click the **Setup** cell in the Linear Buckling system and select **Edit**, or double-clicking the **Setup** cell to open the Mechanical application. In the Mechanical application window, set your linear buckling controls using the Mechanical application's tools and features. See Linear Buckling Analysis in the Mechanical application help for more information on conducting a linear buckling analysis in the Mechanical application.
- 4. On the Toolbar, click **Update Project**.

Magnetostatic

Magnetic fields may exist as a result of a current or a permanent magnet. In ANSYS Workbench, you can perform 3-D static magnetic field analysis. You can model various physical regions including iron, air, permanent magnets, and conductors.

You will configure your magnetostatic analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

- 1. Add a magnetostatic analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the Model cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.

4. In the Mechanical application window, complete your magnetostatic analysis using the Mechanical application's tools and features. See Magnetostatic Analysis in the Mechanical application help for more information on conducting a magnetostatic analysis in the Mechanical application.

Modal and Modal (Samcef)

A modal analysis determines the vibration characteristics (natural frequencies and corresponding mode shapes) of a structure or a machine component. It can serve as a starting point for other types of analyses by detecting unconstrained bodies in a contact analysis or by indicating the necessary time-step size for a transient analysis, for example. In addition, the modal-analysis results may be used in a downstream dynamic simulation employing mode-superposition methods, such as a harmonic response analysis, a random vibration analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions.

You will configure your modal analysis in the Mechanical application, which uses either the ANSYS or Samcef solver, depending on which system you selected, to compute the solution.

- 1. Add a modal analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- View the geometry by right-clicking on the Model cell and choosing Edit, or double-clicking the Model
 cell. Alternatively, you can right click the Setup cell and select Edit. This step will launch the Mechanical
 application.
- 4. In the Mechanical application window, complete your modal analysis using the Mechanical application's tools and features. See Modal Analysis in the Mechanical application help for more information on conducting a modal analysis in the Mechanical application.

Random Vibration

The loads on a structural system may not always be known or quantifiable with certainty. Sensitive electronic equipment mounted in an automobile, for example, may experience slightly-different conditions each day, due to changes in engine vibration or pavement roughness, even if the same road is traveled. A random vibration analysis enables you to determine the response of structures to vibration loads that are random in nature. Since the input loads are described using statistical quantities, the outputs are said to be nondeterministic, meaning that the results can only be known within a certain probability.

You will configure your random vibration analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

A random vibration analysis must follow a modal analysis that extracts the natural frequencies and mode shape. See the instructions in Modal and Modal (Samcef) (p. 145) to build a Modal analysis system. Then follow the instructions below. Alternatively, you can select (double-click) Random Vibration from Custom Systems in the Toolbox. This option creates a random vibration system template that includes both the modal analysis and random vibration analysis system templates with the links pre-defined.

- From the modal analysis system, right click on the Solution cell and select Transfer Data to New>Random Vibration.
- 2. A new random vibration system is created, with the **Engineering Data**, **Geometry**, **Model**, and **Setup** cells linked from the modal analysis system.

- 3. Right click the **Setup** cell in the random vibration system and select **Edit**, or double-click the **Setup** cell to open the Mechanical application. In the Mechanical application window, set your random vibration controls using the Mechanical application's tools and features. See Random Vibration Analysis in the Mechanical application help for more information on conducting a random vibration analysis in the Mechanical application.
- 4. On the Toolbar, click **Update Project**.

Response Spectrum

A response spectrum analysis has similarities to a random vibration analysis. However, unlike a random vibration analysis, responses from a response spectrum analysis are deterministic maxima. For a given excitation, the maximum response is calculated based upon the input response spectrum and the method used to combine the modal responses. The combination methods available are: the Square Root of the Sum of the Squares (SRSS), the Complete Quadratic Combination (CQC) and the Rosenblueth's Double Sum Combination (ROSE).

You will configure your response spectrum analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

A response spectrum analysis must follow a modal analysis. See the instructions in Modal and Modal (Samcef) (p. 145) to build a Modal analysis system. Then follow the instructions below. Alternatively, you can select (double-click) Response Spectrum from Custom Systems in the Toolbox. This option creates a response spectrum system template that includes both the modal analysis and response spectrum analysis system templates with the links pre-defined.

- 1. From the modal analysis system, right click on the **Solution** cell and select **Transfer Data to New>Response Spectrum**.
- 2. A new response spectrum system is created, with the **Engineering Data**, **Geometry**, **Model**, and **Setup** cells linked to the modal analysis system.
- 3. Right click the **Setup** cell in the response spectrum system and select **Edit**, or double-click the **Setup** cell to open the Mechanical application. In the Mechanical application window, set your response spectrum controls using the Mechanical application's tools and features. See Response Spectrum Analysis in the Mechanical application help for more information on conducting a response spectrum analysis in the Mechanical application.
- 4. On the Toolbar, click **Update Project**.

Rigid Dynamics

You can perform a rigid dynamic analysis that specifically uses the ANSYS Rigid Dynamics solver. This type of analysis is used to determine the dynamic response of an assembly of rigid bodies linked by joints and springs. You can use this type of analysis to study the kinematics of a robot arm or a crankshaft system for example.

You will configure your rigid dynamics analysis in the Mechanical application, which uses the ANSYS Rigid Dynamics solver to compute the solution.

- 1. Add a Rigid Dynamics analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry.

- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the **Model** cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.
- 4. In the Mechanical application window, complete your transient structural analysis using the Mechanical application's tools and features. See Rigid Dynamics Analysis in the Mechanical application help for more information on conducting a Rigid Dynamics analysis in the Mechanical application.

Static Structural and Static Structural (Samcef)

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

You will configure your static structural analysis in the Mechanical application, which uses the ANSYS or Samcef solver, depending on which system you selected, to compute the solution.

- 1. Add a static structural analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- View the geometry by right-clicking on the Model cell and choosing Edit, or double-clicking the Model
 cell. Alternatively, you can right click the Setup cell and select Edit. This step will launch the Mechanical
 application.
- 4. In the Mechanical application window, complete your static structural analysis using the Mechanical application's tools and features. See Static Structural Analysis in the Mechanical application help for more information on conducting a structural analysis in the Mechanical application.

Steady-State Thermal and Steady-State Thermal (Samcef)

You can use a steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineers often perform a steady-state analysis before performing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

You will configure your steady-state thermal analysis in the Mechanical application, which uses the ANSYS or Samcef solver to compute the solution.

- 1. Add a steady-state thermal analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- View the geometry by right-clicking on the Model cell and choosing Edit, or double-clicking the Model
 cell. Alternatively, you can right click the Setup cell and select Edit. This step will launch the Mechanical
 application.
- 4. In the Mechanical application window, complete your steady-state thermal analysis using the Mechanical application's tools and features. See Steady-State Thermal Analysis in the Mechanical application

help for more information on conducting a steady-state thermal analysis in the Mechanical application. See the *Thermal Analysis Guide* for more information on thermal analyses using the Mechanical APDL application.

Thermal-Electric

A Steady-State Thermal-Electric Conduction analysis allows for a simultaneous solution of thermal and electric fields. This coupled-field capability models joule heating for resistive materials as well as Seebeck, Peltier, and Thomson effects for thermoelectricity.

You will configure your thermal-electric analysis in the Mechanical application, which uses the ANSYS solver to compute the solution.

- 1. Add a thermal-electric analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.

3. Setup

View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the **Model** cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.

4. In the Mechanical application window, complete your thermal-electric analysis using the Mechanical application's tools and features. See Thermal-Electric Analysis in the Mechanical application help for more information on conducting a thermal-electric analysis in the Mechanical application.

Throughflow

The Throughflow analysis system contains cells for conducting a study with Vista TF starting with the geometry and ending with a report. It is essentially a Vista TF system with an added **Geometry** cell. For details on the Vista TF system, see Vista TF (p. 239).

Transient Structural and Transient Structural (Samcef)

You can perform a transient structural analysis (also called time-history analysis) that specifically uses the ANSYS Mechanical solver. This type of analysis is used to determine the dynamic response of a structure under the action of any general time-dependent loads. You can use it to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any transient loads. The time scale of the loading is such that the inertia or damping effects are considered to be important. If the inertia and damping effects are not important, you might be able to use a static analysis instead.

You will configure your Transient Structural analysis in the Mechanical application, which uses the ANSYS Mechanical or Samcef solver to compute the solution.

- 1. Add a Transient Structural analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry.
- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the **Model** cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.

4. In the Mechanical application window, complete your transient structural analysis using the Mechanical application's tools and features. See Transient Structural Analysis in the Mechanical application help for more information on conducting a transient structural analysis in the Mechanical application.

Transient Thermal and Transient Thermal (Samcef)

Transient thermal analyses determine temperatures and other thermal quantities that vary over time. The variation of temperature distribution over time is of interest in many applications such as with cooling of electronic packages or a quenching analysis for heat treatment. Also of interest are the temperature distribution results in thermal stresses that can cause failure. In such cases the temperatures from a transient thermal analysis are used as inputs to a static structural analysis for thermal stress evaluations. Many heat transfer applications such as heat treatment problems, electronic package design, nozzles, engine blocks, pressure vessels, etc., involve transient thermal analyses.

You will configure your transient thermal analysis in the Mechanical application, which uses the ANSYS or Samcef solver to compute the solution.

- 1. Add a transient thermal analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Load the geometry by right-clicking on the **Geometry** cell and choosing **Import Geometry**.
- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**, or double-clicking the **Model** cell. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.
- 4. In the Mechanical application window, complete your transient thermal analysis using the Mechanical application's tools and features. See Transient Thermal Analysis in the Mechanical application help for more information on conducting a transient thermal analysis in the Mechanical application. See the *Thermal Analysis Guide* for more information on thermal analyses using the Mechanical APDL application; specifically, for more information on time stepping, see the discussion on general load step options.

Component Systems

Component systems allow you to launch and use familiar standalone editors to build a project. They usually do not include all components and steps necessary to achieve an analysis system; however, if you are familiar with a particular application, you can use your product knowledge to complete an analysis.

Component systems available with ANSYS Workbench include the following. Systems will be available only if you have installed the product and have an appropriate license.

Autodyn BladeGen

CFX

Engineering Data

Explicit Dynamics (LS-DYNA Export)

External Data

External Model

Finite Element Modeler

Fluent

Fluent (with TGrid meshing)

Geometry

ANSYS ICEM CFD

Icepak

Mechanical APDL

Mechanical Model

Mesh

Microsoft Office Excel

Polyflow

Results

System Coupling

TurboGrid

Vista AFD

Vista CCD and Vista CCD with CCM

Vista CPD

Vista RTD

Vista TF

When you double-click or drag a component system onto the Project Schematic, you will see that type of system.

Autodyn

You can use the Autodyn system to launch the standalone Autodyn application. This application supports the full range of Autodyn capabilities including the Explicit Eulerian solvers, meshfree SPH solvers, and explicit solver coupling (FSI).

Note

The Explicit Dynamics analysis system supports the FE components of the Autodyn solver.

Input Files: The Autodyn system takes the binary database (.ad) file as input.

Output Files: Files that are produced by all Autodyn runs. Primary output files include:

- results file (.adres) for post-process
- save file (admodel cycle.adres) for post-processing and database modification during a run
- history data files (.his, .sum) that record time dependant data at gauge locations and summary data for materials/parts
- print file (.prt) recording a text summary of the model definition and results
- log file (.log) listing solution information and warnings or errors encountered

Working with an Autodyn Analysis

To add an Autodyn analysis to your project, double-click the Autodyn object or drag-and-drop it from the Component Systems area of the Toolbox into the Project Schematic. An Autodyn system appears in the Project Schematic.

You can connect other systems to an Autodyn system by using the **Transfer Data From New** context menu options. When transferring data to an Autodyn system from another system, you can transfer data to the **Setup** cell from two locations:

- **Setup** cell of an analysis system (most commonly an Explicit Dynamics system)
- Mesh cell of a Mesh component system

In both cases, you must update the **Setup/Mesh** cell after connecting to the Autodyn **Setup** cell. This update operation produces a CAERep. xml file that can subsequently be consumed by the Autodyn **Setup** cell.

To launch Autodyn interactively, right mouse-click and select **Edit Model** or **New Model**.

To launch Autodyn with input and reference files specified, right mouse-click the **Setup** cell and select **Import Model**. Then select **Edit Model**. Autodyn will launch in interactive mode, and the input file(s) specified will be loaded.

ANSYS Autodyn Context Menu Options

The Autodyn system contains two cells:

- 1. **Setup**: **Setup** cell context menu options include the following.
 - **New Model**: Opens the Autodyn editor where you can set up a new 2-D or 3-D Autodyn model, solve, or post-process results.
 - **Edit Model**: Opens the Autodyn editor and loads in the database currently associated with system. You can then further edit the model, solve, or postprocess results.
 - **Import Model**: Imports an existing Autodyn database (.ad file) into the system. This action will replace any existing database associated with the system. If the import detects that there are other files in the source directory for the model that is being imported the user will be presented with the option to import all the associated files.
 - **Select User Executable**: Selects the Autodyn executable file (autodyn.exe) you want to associate with the system and use for subsequent preprocessing, solving, and postprocessing. Typically this option is used to select a user-customized executable.
 - Transfer New Data From
 - Mesh: Inserts a Mesh component system and generates a link between its Mesh cell and the Autodyn system Setup cell. This option enables the transfer of a mesh from the ANSYS Meshing system into the Autodyn system.
 - Explicit Dynamics: Inserts an Explicit Dynamics analysis system and generates a link between its
 Setup cell and the Autodyn system Setup cell. This option enables the transfer of the initial model defined in the Explicit Dynamics system or the Autodyn system. The initial model includes materials, mesh, connections, coordinate systems, initial conditions, loads, constraints, and analysis settings.
 - **Update**: Update is used to transfer the latest upstream data from the Mesh or Explicit Dynamics system into the Autodyn system.

Note

Any modifications made in Autodyn to a model that originated from an Explicit Dynamics system are likely to be overwritten during the update process. Items defined in the Explicit Dynamics or Mesh system represent the master version of the data.

- **Duplicate**, **Properties**, **Rename**: Standard actions as described in Cells in Workbench (p. 252).
- 2. Analysis: Analysis cell context menu items include the following.
 - **Duplicate**: Duplicates the Autodyn system. All data associated with the system (including results files) will be copied into a new Autodyn system.
 - Update, Clear Generated Data, Rename: Standard actions as described in Cells in Workbench (p. 252).

BladeGen

BladeGen is a component of ANSYS BladeModeler. The BladeModeler software is a specialized, easy-to-use tool for the rapid 3-D design of rotating machinery components. Incorporating extensive turbomachinery expertise into a user-friendly graphical environment, the software can be used to design axial, mixed-flow and radial blade components in applications such as pumps, compressors, fans, blowers, turbines, expanders, turbochargers, inducers and others.

BladeModeler provides the essential link between blade design and advanced simulation including computational fluid dynamics and stress analyses. BladeModeler contains a rich set of tools and functions for designing a turbomachinery blade from scratch, using industry-specific tools, workflow, and language that the blade designer expects.

With BladeGen, the user can re-design existing blades to achieve new design goals or create completely new blade designs from scratch. When either re-designing or evaluating an existing blade design, BladeGen facilitates the import of blade geometry interactively or through user supplied files. BladeGen allows sculpted or ruled element blades with linear or compound lean leading or trailing edges. Over/Under-Filing can be applied and leading and trailing edge shapes are easily specified as a full radius, an ellipse ratio, or a simple cutoff.

BladeModeler represents a pivotal link between blade design, advanced analysis and manufacturing. Used in combination with ANSYS analysis software, users can rapidly evaluate the performance of a component. BladeGen model files can be imported into DesignModeler using the BladeEditor feature. BladeEditor provides a seamless path to both structural and fluid analysis, which enables the user to efficiently transition from preliminary blade design, to full 3-D viscous flow analysis, and finally to the users native CAD system.

To run BladeGen, drag the BladeGen component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox, then edit the **Blade Design** cell. For more information on ANSYS BladeGen, see the ANSYS BladeGen help.

To run BladeEditor, drag the Geometry component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox. Edit the **Geometry** cell to invoke DesignModeler. Inside DesignModeler, BladeEditor features will be available depending on your license preferences. To set the license preferences for running BladeEditor, follow the instructions in Configuring the ANSYS BladeModeler License. For more information on ANSYS BladeEditor, see ANSYS BladeEditor help.

CFX

ANSYS CFX enables you to perform fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You import meshes, specify the materials, boundary conditions, and solution parameters, solve the calculations, view the results, then create reports using built-in tools.

To use ANSYS CFX as a component system in ANSYS Workbench:

- 1. Add a CFX component system by dragging the system from the Component System Toolbox into the Project Schematic or by double-clicking the system in the Toolbox.
- Read a mesh and specify the ANSYS CFX physics definitions in CFX-Pre by right-clicking on the Setup
 cell and choosing Edit. Alternatively, you can import a previously saved case file by right-clicking on
 the Setup cell and choosing Import Case > Browse.
- 3. Right-click on the **Solution** cell and choose **Update** to start the solver. Alternatively, right-click on the **Solution** cell and select **Edit**, set the solver execution controls in CFX-Solver Manager, and start the solver.
 - You can also import an existing CFX-Solver Results file by right-clicking on the **Solution** cell and choosing **Import Solution** > **Browse**.
- 4. Analyze the results of the calculations in CFD-Post by right-clicking on the **Results** cell and choosing **Edit**.

For detailed information on working with ANSYS CFX, see ANSYS CFX in ANSYS Workbench in the *CFX Introduction*.

For information on submitting ANSYS CFX jobs to Remote Solve Manager, see Submitting CFX Jobs to RSM.

Engineering Data

Use the **Engineering Data** cell with the Mechanical application systems or the Engineering Data component system to define or access material models for use in an analysis.

To add an **Engineering Data** component system to the Project Schematic, drag the Engineering Data component system from the Toolbox to the Project Schematic or double-click the system in the Toolbox.

Double-click the **Engineering Data** cell, or right-mouse click and choose **Edit** from the context menu to display the Engineering Data tab, where you can add or modify material data.

For detailed information on working with Engineering Data, see Engineering Data.

Explicit Dynamics (LS-DYNA Export)

- 1. Add an**LS-DYNA** explicit dynamics analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 2. Add an**LS-DYNA** explicit dynamics analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.
- 3. View the geometry by right-clicking on the **Model** cell and choosing **Edit**. Alternatively, you can right click the **Setup** cell and select **Edit**. This step will launch the Mechanical application.
- 4. In the Mechanical application window, complete your explicit dynamics analysis setup using the Mechanical application's tools and features. See Explicit Dynamics Analysis in the Mechanical application help for more information on exporting to LS-DYNA using an explicit dynamics analysis in the Mechanical application.

If you have a Mechanical APDL system linked to the LS-DYNA system, and you attempt to launch Mechanical APDL using an ANSYS LS-DYNA license (commercial or academic), you may see the following error in the Mechanical APDL output window:

***FATAL

***Parallel capability is not valid for this product

If you see this message, set the number of processors for Mechanical APDL to 1 (**Tools> Options> Mechanical APDL**). You will then be able to run Mechanical APDL and solve an ANSYS LS-DYNA analysis.

External Data

The **External Data** system enables you to import data from text files and feed that data into a Mechanical application or a System Coupling component system. You need to specify the data format in order to process the files in the **External Data** tab. This information can then be transferred to a downstream Mechanical application where the data can be applied as loads in an analysis.

Note

If a file imported into **External Data** is updated and you want systems connected to external data to use the updated data, then you must manually re-read the data and update the **External Data** and downstream systems. Care is consequently required when attempting to use parameters and design points with projects that include **External Data** systems. Specifically, these systems will not automatically re-read imported files or be updated as parameters and design points are updated.

Creating and Configuring an External Data System

To create an **External Data** system:

- 1. Drag an **External Data** system from the **Component Systems** Toolbox onto the Project Schematic.
- 2. To display the **External Data** tab, double-click the **Setup** cell, or right-click and choose **Edit** from the context menu.

You can now add the files in the Outline view.

- 3. To add files:
 - a. In the **Location** column, you may browse to local files using the **Browse** option or to files stored on an EKM repository using the **Browse from Repository** option. For more information on **Browse from Repository**, see Importing Repository Files (p. 131).

When you click **Open**, the selected file names, locations, and identifiers are automatically displayed in the **Data Source** column. You can enter descriptions for the files in the **Description** column.

Table 4: Data Source View: Definition Section

Property	Description			
Identifier	A string that can be used to identify the file in the downstream Mechanical application.			
Master	Enables you to choose a master file so that the system displays and uses the Properties for this file only. The nodal coordinate data for the selected master file will be processed by the Mapping Utility. Any nodal coordinate data contained in non-master files will be skipped.			

Property	Description				
You can select only one file to be the master file. If you multi-select master file and non-master files, the Dimension, Length Unit and T formation properties in the Properties > Definition view will be a lf you duplicate the master file, only one instance will be designated the master file.					
Descrip- tion	Text that describes the file to you and to other users.				

- b. Optionally, you can right-click a file (or files) in the Outline view and use the context menu to duplicate them.
 - All files—whether imported or duplicated—can be sorted or filtered.
- c. Once the files have been added, use the **Properties** view to input the information required to process the file and apply the data in the Mechanical application.

If you select multiple files in the **Data Source** column, the **Properties** view displays:

- A value when that value is the same for all selected files.
- · A blank field when values differ between selected files
- A yellow field when a value is required, but not currently specified for at least one of the files.

If you edit any field in the **Properties** view when multiple files are selected, your change is applied to all files.

Note

Although you can multi-select files in the **Data Source** view, when you click away from that view the highlighting applied to those files disappears. However, the files remain selected and operations subsequently performed will be applied to all of the selected files.

Table 5: Properties View: Definition Section

Property	Description
Dimension	You can choose to either import data from 2D or 3D models. If the 2D option is selected, you will be able to import data only at the X and Y coordinates. The Z coordinate is not supported for the 2D option.
Start Import at Line	The line number at which you want the data import to start. Line numbers start at 1.
Format	 Delimited causes the Delimiter Character field to appear, which enables you to specify the character that delimits data elements in a line. The options are Comma, Semicolon, Space, Tab, or User-Defined. The latter choice enables you to specify any character as the data delimiter.

Description				
 User-Defined enables you to specify the format specification for the file. The allowed format specifiers are given below.^a 				
Note				
For release 14.5, the default Format Type has changed to Delimited, with a Delimited Type of comma.				
A second, related change has been made to the way that column data is numbered. Prior to release 14.5, data was numbered consecutively for each system, with numbers for new data always being incremented from the last number in the file. If any data was removed, the associated numbers for that system would be missing from the file even if new data was added. As of release 14.5, new data is numbered so that whenever possible there are no gaps in the data numbers in the file. This change may break scripts that read column data numbers. Any file using the Delimited format type, must not contain the delimited character(s) at the beginning of each line.				
The unit system in which source point locations are defined.				
Specifies the nature of the source point locations. Options include Cartesian (X,Y,Z) or Cylindrical (R,theta,Z). The default value is Cartesian.				
If Yes, the data at Midside nodes, if not specified, is calculated as average of data specified at corner nodes.				
Note				
This property is only available when Format Type is Delimited or User Defined and cdb file is chosen as Master .				

^aThe format specification used here is drawn from the C format specification.

Table 6: Properties View: Analytical Transformation Section

Property	Description
X/Y/Z Coordin- ate	Enables you to apply scaling factors or functions to the corresponding column Data Type selected in the Table of File view. For example, to scale all values for the X Coordinate column by 90%, change the X Coordinate text entry to x*0.9.
	When the Coordinate System Type is cylindrical, the X Coordinate will refer to the radius and the Y Coordinate refers to the angle. By default, each X, Y, and Z Coordinate is set to x, y, and z.

Property	Description				
	For a complete list of supported functions, see Parameters. For an example, see Source Point Analytical Transformations (p. 168).				
	Note				
	When the Dimension type is 2D, the Z Coordinate is not shown.				
	Rotations, resulting from specified analytical transformations, do not get applied to mapped data (pressure, displacement, force) in a downstream Mechanical system.				

Table 7: Properties View: Rigid Transformation Section

Property	Description
Origin X/Y/Z and Theta XY/YZ/ZX	Enables you to apply a coordinate transformation to the source points. The source locations are transformed by the coordinate system defined by the Origin and Theta entries. For example, applying an X value of .1 meters would modify the x locations of all the source points by adding .1 meters to their values. The option Display Source Points on an imported load object inside Mechanical respects this transformation and can be very helpful in assuring proper alignment between the source and target points. This option is useful if the source points are defined with respect to a coordinate system that is not aligned with the target geometry system.
	Rotation transformations (Theta XY/YZ/ZX) specified in the External Data system will be applied to the mapped data (pressure, displacement, force) in a downstream Mechanical system. Prior to release 14.5, rotation transformations were not applied.

Table 8: Allowed Format Specifiers

Туре	Specifier
Decimal floating point	F or f
Scientific notation (exponent) using E/e character	E or e
Signed decimal in- teger	D or d
Signed integer	l or i
Character	A or a
Ignore flag	X or x

In the example below, the line has one integer followed by four exponential data types.

```
385 8.33333333E-003 9.375000000E-003 3.375000000E-001 0.000000000E+000
```

The corresponding format is

1i3,4e17.9

where:

- The first entry is the number of occurrences
- The second entry is the format specifier (from Table 8: Allowed Format Specifiers (p. 157))
- The third entry is the number of characters of data in the definition (including numeric values (0-9), the letter 'e' (for instances of scientific notation), and any white space and + or signs)
- The fourth entry is the number of digits after the decimal point.

Tip

By default, Excel puts a minus sign at the beginning of a negative number but it does not put a plus sign at the beginning of a positive number, which changes the number of characters in the negative version of a number. However, if you select a column in Excel and apply a custom format, you can have a plus sign added to positive numbers. In Excel's **Format Cells** dialog box, set the Type with an example of the format for a positive number and a negative number, separating the two types with a semicolon. For example:

+00.00E+00;-00.00E+00

- 4. Based on the format details you specified in the **Properties** view, you will need to define the data types and their corresponding units in the **Table of File** view. The allowed **Data Type** values are:
 - · Not Used
 - X Coordinate
 - Y Coordinate
 - Z Coordinate
 - Element ID
 - Node ID
 - Temperature
 - Pressure
 - · Heat Transfer Coefficient
 - Heat Flux

- · Heat Rate
- · Heat Generation
- Thickness
- Displacement
- Force
- Velocity
- Stress
- Strain
- · Body Force Density

Note

Thickness uses the Length **Data Unit** and the **Data Identifier** should start with Thickness.

Tip

You can multi-select rows (from the left-most column in the **Table of File** view), then right-click to set all highlighted Data Type values or Data Unit values (where applicable) at the same time.

Column data is ignored if the **Data Type** is set to Not Used.

When **Coordinate System Type** is set to Cartesian in the **Properties** view, the Data Unit cell for the X, Y, and Z Coordinates will be read-only in the **Table of File** view. When Coordinate System Type is set to Cylindrical, the Data Unit cell for the Y will have a combo box that can specify its Data Unit string (either as Degrees or Radian).

You can change the data identifier from the default string for allowed data types. The data identifiers are appended to the file identifier, specified in the **Properties** view, so that you can pick the correct source data in the downstream Mechanical application.

A preview of the file is shown in the **Preview** view. The first ten imported lines are shown.

When you multi-select files, the **Preview** view is disabled and the **Table of File** view displays data in columns that you can sort and filter from the down arrow beside each column heading.

5. The **Setup** cell of the **External Data** system can be linked to a **Model** cell or to a **Setup** cell of a Mechanical system (except for Rigid Dynamics Systems).

You can modify any file in the **Outline** view by browsing to a new file using the browse option provided in the **Location** column

Note

If you modify an **External Data** system's data file outside of Workbench, you need to cause Workbench to re-read the data file: right-click the **Setup** cell and select **Re-read Input Files**.

Note that the **Re-read Input Files** operation will cause Workbench to regard the file as having changed whether the file has changed or not, and the status of the **Setup** cell will change appropriately.

You can also delete files that you have selected (or multi-selected) by right-clicking one of the files in the **Outline** view and then choosing **Delete** from the context menu.

To add a downstream Mechanical system, either drag a valid analysis system from the Toolbox and drop it on the appropriate **Setup** cell of the **External Data** system, or right-click the **Setup** cell and choose the **Transfer Data To New** context menu option.

For additional information for using the data in a downstream Mechanical application, refer to External Data Import.

Importing a CDB File as Input

To import an Mechanical APDL-generated CDB file as a source file using the **External Data** component, select a file or files from the **Location** field in the **Outline**. Once selected, the **Format Type** property is automatically set to .cdb.

CDB files have certain restrictions:

- The **Start Import at Line** setting is disabled as the entire file is read as data. **Delimiter Type**, **Delimiter Character**, and **Format String** are also disabled.
- The data is always in the Cartesian global frame-of-reference.
- The Table and Preview panes are disabled, so you cannot specify data types or preview data.

Mechanical APDL CDB files can be added as a master mesh in the External Data system. The file must be generated using 'blocked' formatting (see **CDWRITE** in the *Mechanical APDL Command Reference*). Files generated in unblocked format are not supported. Only NBLOCK and EBLOCK data will be read from the file. Elements defined in the EBLOCK command should also have a corresponding element type and number (ET command) defined in the CDB file. No load/data transfer information is read.

The following element types are ignored during reading of the file:

- Solid168
- Targe169,170
- Conta171,172,173,174,175,176,177,178
- SURF152,153,154,156

Data transfer information must be defined in separate files, which will also need to be added to the same External Data system. These files must have a column providing node identifiers that match the node IDs defined in the Mechanical APDL CDB file. If the data transfer file contains element nodal values, then both node and element identifier columns need to be defined. Element values are not supported.

Note

- An element defined in the CDB file will be ignored by the common mapping utility if any of the below conditions exist. The ignored elements will not be used when data is mapped in the downstream application.
 - One or more of the nodes forming the element are not defined in the NBLOCK command.
 - The element does not have a corresponding element type and number (ET command) defined in the CDB file.

Note

- CDB files must contain only solid elements or shell elements, but not both.
- For nodal data transfers, the number of nodes in the Mechanical APDL CDB file must match the number of data transfer items in all slave files.
- If data is not available at midside nodes, then **Average Data at Midside Nodes** property can be used to specify the data at Midside nodes as an average of data at corner nodes.

Importing an ANSYS External Data File as Input

To import an .axdt file as a source file, select **Data Sources** > **Location**. When you specify an ANSYS External Data file, the **Format Type** is automatically set to **ANSYS External Data File**. Also, the **Length Unit** property will not be available because this information is specified in the file. All length unit information in the .axdt file must be the same (all meters, for example).

ANSYS External Data files have the restriction that the **Start Import at Line** setting is disabled as the entire file is read as data. **Delimiter Type**, **Delimiter Character**, and **Format String** are also disabled.

This file format is described in the next section.

ANSYS External Data File Format

CFD-Post and the Mechanical application can export data files (.axdt) that can be used by the External Data system. Note that the Mechanical application does not export temperature data in units of K. The following is an example of an .axdt file that has been exported from CFD-Post:

```
[Name]
Plane 1

[Data]
X [m] (X coordinate), Y [m] (Y coordinate), Z [ m] (Z coordinate),
Wall Heat Transfer Coefficient [W m^-2 K^-1] (Heat Transfer Coefficient),
Wall Adjacent Temperature [K] (Temperature)
-1.77312009e-02, -5.38203605e-02, 6.00000024e-02, 7.12153496e-06, ...
-1.77312009e-02, -5.79627529e-02, 5.99999949e-02, 5.06326614e-06, ...
.
```

```
.
[Faces]
369, 370, 376, 367
350, 374, 367, 368
.
```

This file contains three blocks, each with one of the labels: [Name], [Data] and [Faces]. The [Name] block contains the name of the region contained/defined in the file.

The [Data] block contains node coordinates and values. The first line following the [Data] label is a header that contains a comma separated list of unique labels, units and quantity type for coordinates and values at each node. Units are contained in square brackets and quantity types are contained in parentheses. Subsequent lines, one per node, contain a comma separated list of data defined in the header. The [Data] block ends in the line before the [Faces] block label.

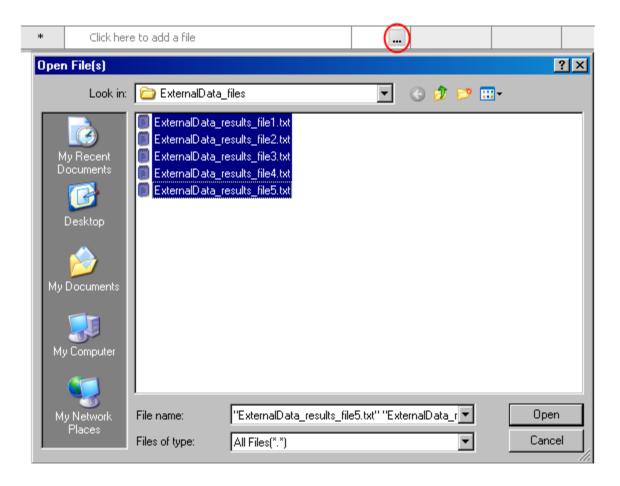
The [Faces] block contains definitions for topologically two-dimensional faces (small surfaces), each by 3 (triangle) or 4 (quadrilateral) points. The points must be ordered to trace a path going around the face. For proper rendering, the faces should have consistent point ordering, either clockwise or counterclockwise. Each face is automatically closed by connecting the last point to the first point. Face connectivity data is listed in the [Faces] block and references the points in the [Data] block, where the latter are implicitly numbered, starting with 0.

Importing Multiple Data Sets

External Data can be configured to efficiently import multiple data sets (for example, from a transient analysis). It can handle multiple sets through a single file or via multiple files.

This section will guide you through the steps to set up such an analysis:

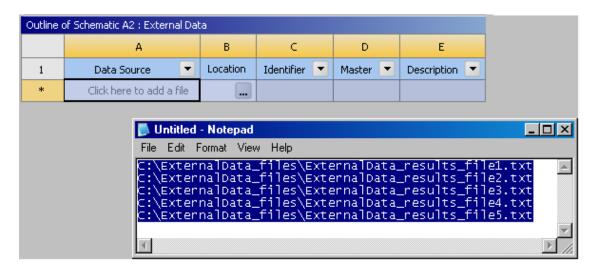
- Create the External Data system by double-clicking External Data in the Component Systems toolbox.
 An External Data system appears in the Project Schematic.
- 2. Double-click the External Data system's **Setup** cell to edit it. The Outline view, Properties view, and Table view appear.
- 3. Using the Outline view, choose the data files. You can:
 - Perform multiple file add operations in the Outline view.
 - In the Location column, click the browse icon (circled in the figure that follows), multi-select files in the **Open File(s)** dialog box that appears, and click **Open**.



• If you have a list of fully-qualified paths to the files you want, you can copy the list from a text file or an Excel file and paste it into the Outline view's Data Source field.

To paste from a flat-text editor such as Notepad:

- 1. List the paths to the files in the editor. Ensure that there are no trailing spaces in the lines.
- 2. Select all the files and copy them (**Ctrl+A**, then **Ctrl+C**)
- 3. In the Outline view, click the asterisk. The line becomes highlighted, and the text **Click here to add a file** *remains visible*.



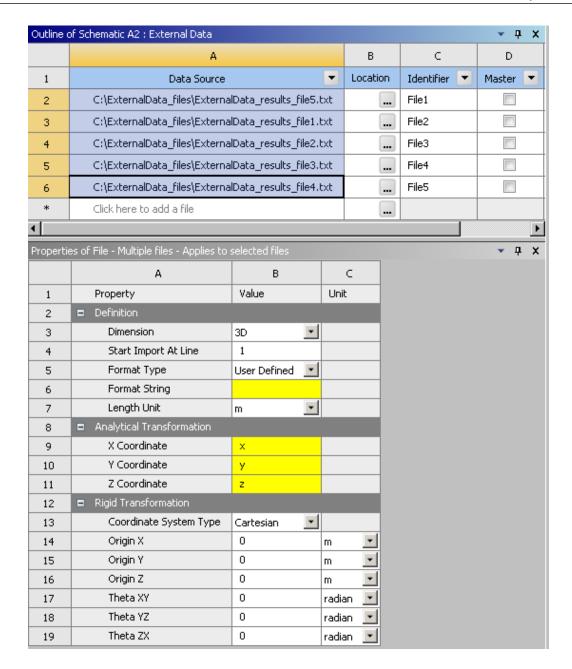
4. Press Ctrl+V to add the files.

Outline o	Outline of Schematic A2 : External Data						
	А	В	С	D			
1	Data Source	Location	Identifier 💌	Master 💌	Descrip		
2	C:\ExternalData_files\ExternalData_results_file1.txt		File1				
3	C:\ExternalData_files\ExternalData_results_file2.txt		File2				
4	C:\ExternalData_files\ExternalData_results_file3.txt		File3				
5	C:\ExternalData_files\ExternalData_results_file4.txt		File4				
6	C:\ExternalData_files\ExternalData_results_file5.txt		File5				
*	Click here to add a file						

4. In the Properties view, set the properties of the files.

If the files are of the same or similar format, you can make use of multi-selection of the files for quick settings of common properties:

- a. Multi-select the desired files in the Outline view. You will see:
 - A value when that value is the same for all selected files.
 - A blank field when values differ between selected files.
 - A yellow field when a value is required, but is not currently specified for any of the files.



If you edit any field in the Properties view when multiple files are selected, your change is applied to all files.

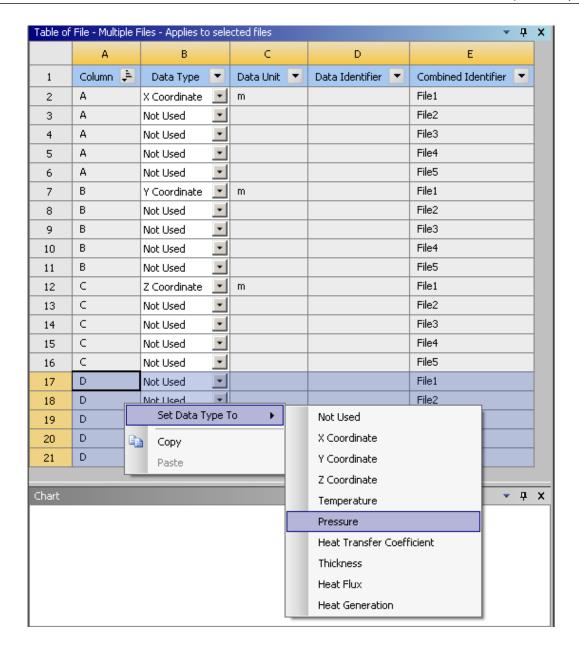
- b. If the X, Y, Z locations of the source points is common between all the files, you can make use of the "Master" Designation. By designating a "Master" file, all other files will use that file's values for the X, Y, Z locations. This leads to faster user interface set up, as well as much faster mapping times as the mapping weight calculations need to be done only once and then are shared for all slave files.
- 5. Use the Table view (which is populated from the **Format String** field in the Properties view) to specify the Column data in the file.

Here again you can make use of multi-selection of the files in order to fully populate the Table view. You can span data from all selected files and use various right-mouse button actions to efficiently define the column data.

- a. In the Outline view, multi-select the desired files.
- b. In the Table view, sort the table by **Column** to efficiently order the file data. For example, if you have four data fields and the first three specify the X, Y, Z locations, sorting by column will place the remaining data-field entries together at the bottom of the column.
- c. Select all the rows for which you want to change data:
 - i. Select the first row to be changed by clicking on the *row number* (which is in the table boundary).
 - ii. Press and hold Shift key.
 - iii. Select the last row to be changed by clicking on the row number.
 - iv. Right-click anywhere over the selected cells, choose **Set Data Type To**, and set the desired data type. Repeat as required to set the data units via **Set Unit To**.

Tip

If **Set Data Type To** or **Set Unit To** are not available, ensure that your mouse cursor is over the body of the table, not on the table boundary.



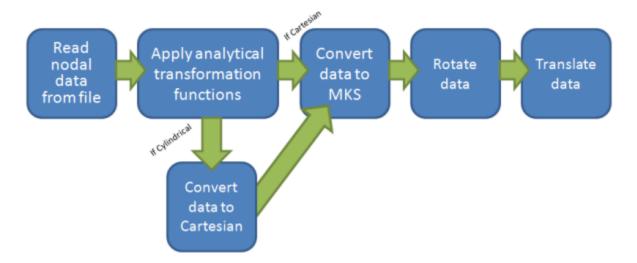
- 6. Select and copy the cell entries in the **Combined Identifier** Column that correspond to the multiple data sets (using **Ctrl+C** or right-click and select **Copy**). These data-identifier strings will be used to specify which data set will be imported at each load step inside Mechanical.
- 7. Link the External Data system into the desired Mechanical system/cell.
- 8. Update the External Data System.
- 9. Edit (or, if editor is already open, Refresh) the Mechanical System.
- 10. Set the desired **Number Of Steps** in Mechanical's **Analysis Settings** object. Set the step end times as desired (you can copy and paste).
- 11. As needed, create the desired Imported Load/Thickness in Mechanical.

- 12. Select the Imported Object, then paste the data identifier text into the appropriate cells inside the **Data** view on the imported object.
- 13. As required, copy and paste the desired step end times on the Analysis Settings Object and the Imported Data Object (in the Analysis Time(s) column).
- 14. Right-click **Import Load** to invoke the mapping calculations.
- 15. After mapping has completed, you can review the various mappings by adjusting the **Active Row** entry in the details view. The graphics will render a contour plot of the imported data at the specified row. Additionally, this data can be exported out of Mechanical by right-clicking the tree object and selecting **Export**.

Source Point Analytical Transformations

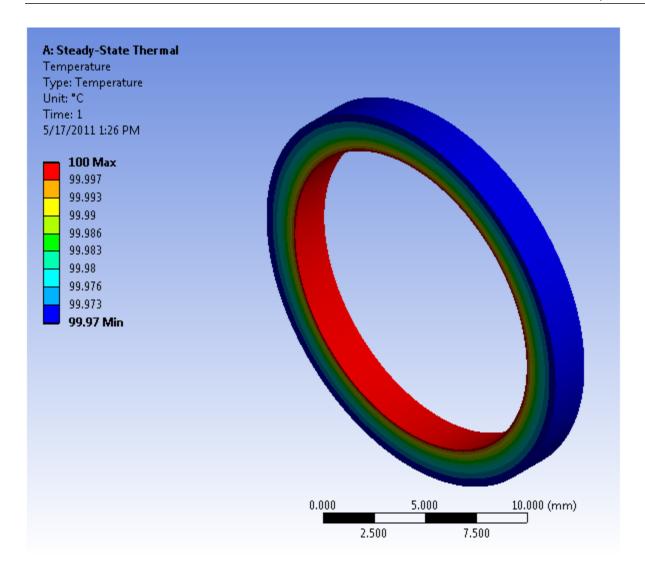
Analytical transformation options can be applied to the source nodal locations. You can enter constant or functional values that will be applied to the x, y, and z (or r, theta, z for cylindrical) values read from the input file from within the External Data User Interface. For a complete list of supported functions, go to Expressions, Quantities, and Units (p. 98). The order of operations for conversion of the original node locations into a format the common mapping utility uses must be taken into consideration when setting up analytical transformation functions, rotation, and translation information.

The nodal data, as well as any unit system information, is read into the common mapping utility. If any nodal analytical transformation values or functions are provided, they are applied directly to the nodal coordinates as they are read in from the file. If the data is provided in a cylindrical system, it is converted into Cartesian coordinates. Once the nodes are in Cartesian, all nodal data is converted into MKS, so that the mapping utility stores all data in the same unit system. If any rotational information is provided, this is applied next, followed by any translations.



The following example takes nodal data, written in a cylindrical system, from a ring with an inner radius of 8 mm and an outer radius of 10 mm and allows an analytical transformation value to be applied to the radius of the source data, such that it aligns with a smaller ring with an inner radius of 7.2 mm and an outer radius of 9 mm.

Thermal results on an expanded or stretched ring (inner radius is 8 mm and outer radius is 10 mm)

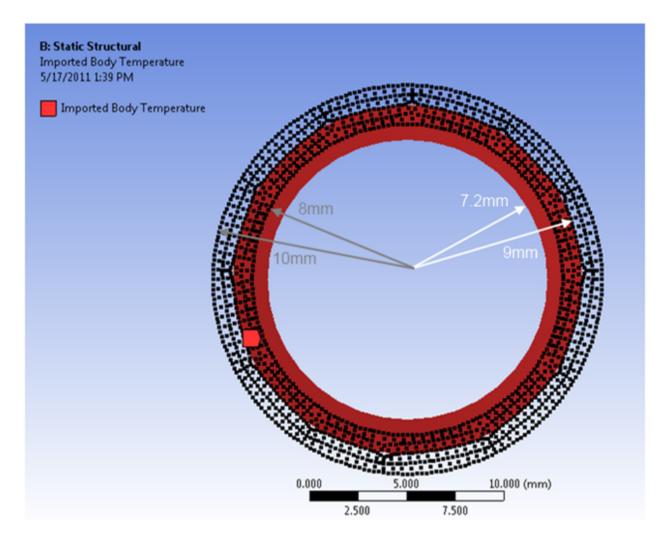


Sample cylindrical nodal data:

Node Number	Radius (mm)	Theta (radi- ans)	Z Location (mm)	Temperature (-C)
1	8.914523065	-0.247535105	1.6	99.983
2	8.914523065	-0.247535105	1.2	99.984
3	8.914523065	-0.247535105	0.8	99.984
4	8.914523065	-0.247535105	0.4	99.983
5	8.917873803	-0.192647608	1.6	99.983
6	8.917873803	-0.192647608	1.2	99.984
7	8.917873803	-0.192647608	0.8	99.984
8	8.917873803	-0.192647608	0.4	99.983
9	8.927166575	-0.137916029	1.6	99.983

If we simply import the nodal data, you can see how the source nodes are not contained within the target volume.

Imported temperature load with unmodified source points displayed:



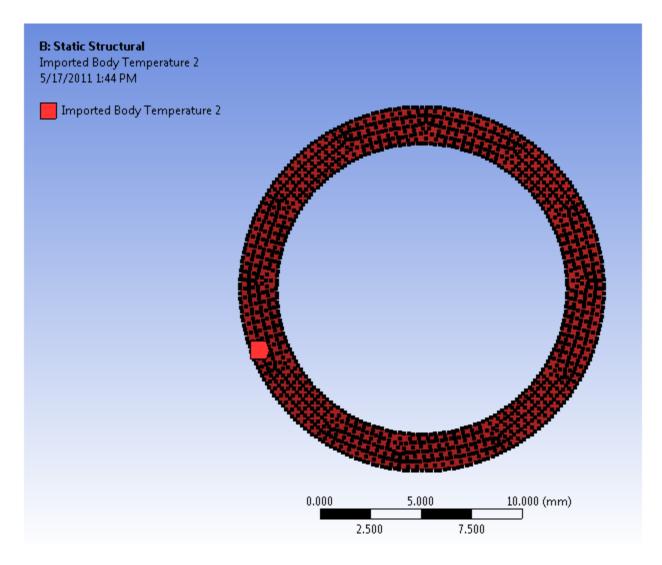
Within the External Data System User Interface, you can enter analytical transformation values as either constants, or as functions of x, y, or z. For our example, we want to scale the source radius (that is, X Coordinate) by 90%.

Applying a constant analytical transformation scale factor to the radius (that is, X Coordinate) of the cylindrical source nodal locations:

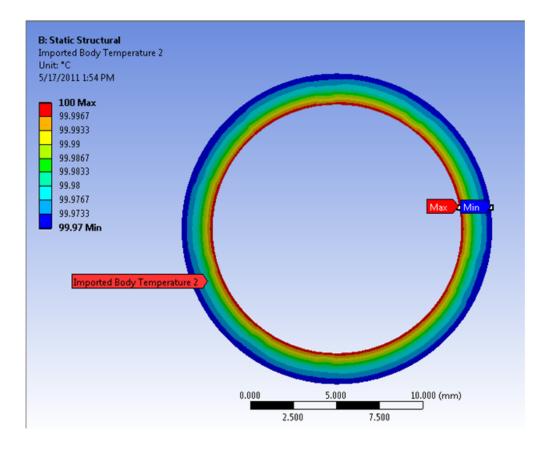
	А	В	С	
1	Property	Value	Unit	
2	■ Definition			
3	Dimension	3D 💌		
4	Start Import At Line	2		
5	Format Type	Delimited 💌		
6	Delimiter Type	Tab 🔽		
7	Delimiter Character	Tab		
8	Length Unit	mm 💌		
9	Coordinate System Type	Cylindrical 🔽		
10	 Analytical Transformation 			
11	X Coordinate	x*.9		
12	Y Coordinate	У		
13	Z Coordinate	z		
14	■ Rigid Transformation			
15	Origin X	0	m 🔽	
16	Origin Y	2	mm 🔽	
17	Origin Z	0	mm 💌	
18	Theta XY	0	radian 🔽	
19	Theta YZ	90	degree 🔽	
20	Theta ZX	0	radian 💌	

Updating the schematic and looking at the imported load we see the analytical transformation value has been applied to the nodal locations and now coincides with the target model.

Imported temperature load with "transformed" source points displayed:



Importing the load generates the following imported load:



Performing System Coupling Simulations Using External Data

You can use Workbench to perform coupled simulations using multiple Analysis or Component Systems. The External Data System may be used as a source of static (that is, unchanging) data for other cosimulation participants such as ANSYS Fluent or ANSYS Mechanical, as described in the System Coupling Guide.

The tutorial Tutorial: Heat Transfer from a Heating Coil in the *System Coupling User's Guide* is an example of a coupled analysis using External Data with System Coupling.

External Data System and System Coupling Configuration

Each External Data system that is connected to the System Coupling system is represented as a coupling participant in the System Coupling setup. The name of this participant will be the name of the External Data system specified in the Workbench Project Schematic. Note that since the External Data system acts as a source of static data, it can only be defined as the source in System Coupling's Data Transfer definition.

Each input file added to an instance of the External Data system is represented as a coupling region for the associated coupling participant in the System Coupling setup.

Supported Input File Formats

In order to use the External Data system as a coupling participant, the "ANSYS External Data File" file format should be used. For information regarding use of this file format, see Importing an ANSYS External Data File as Input (p. 161).

Supported Data Types

Data Types provided by External Data that are currently consumable by the System Coupling system include Temperature and Heat Rate.

Note that in the XML file transferred from the External Data System to System Coupling, if the units of the coordinates in are not all consistent, an error will be reported.

External Model

The **External Model** system enables you to import solid and shell finite element meshes into Workbench. These meshes can then be imported directly into the Mechanical application. When used in this manner, geometry is automatically synthesized from specified meshes and made available inside the Mechanical application. The imported files must have been generated in the Mechanical APDL application in common database (.cdb) format.

See the Mesh-Based Geometry section in the Mechanical help for the file requirements for using this system.

Note

If a file imported into **External Model** tool is updated and you want systems connected to **External Model** to use the data, then you must manually re-read the data and update the **External Model** and any downstream systems. Consequently, you must use care when attempting to use parameters and design points with projects that include **External Model** systems. Specifically, these systems will not automatically re-read imported files or be updated as parameters and design points are updated.

Associativity between External Model and Mechanical

When making updates to a model using the **External Model** component in Workbench, all geometry scoping on objects (such as loads, results, etc.) is lost in the downstream environment. Therefore, you should use either imported named selections or criteria-based named selections to scope the downstream objects. These scoping features automatically update when the upstream model updating is complete.

Creating and Configuring an External Model System

To create an **External Model** system:

- 1. Drag an **External Model** system from the **Component Systems** Toolbox onto the Project Schematic.
- To display the External Model tab, double-click the Setup cell, or right-click and choose Edit from the context menu.

You can now add the files in the **Outline** view.

- 3. To add files:
 - a. In the **Location** column, you may browse to local files using the **Browse** option or to files stored on an EKM repository using the **Browse from Repository** option. For more information on **Browse from Repository**, see Importing Repository Files (p. 131).

When you click **Open**, the selected file names and locations are automatically displayed in the **Data Source** column. You can enter descriptions for the files in the **Description** column.

b. Optionally, you can right-click a file (or files) in the **Outline** view and use the context menu to duplicate them.

All files (whether imported or duplicated) can be sorted or filtered.

- c. Once you have opened your files, use the **Properties** window to modify the unit system and/or coordinate system transformation properties. These properties transform the mesh coordinate systems of the sub-assemblies for proper alignment in Mechanical.
- d. If you select multiple files in the **Data Source** column, the **Properties** view displays:
 - · A value when that value is the same for all selected files
 - A blank field when values differ between selected files
 - A yellow field when a value is required, but not currently specified for at least one of the files.

If you edit any field in the **Properties** view when multiple files are selected, your change is applied to all files.

Caution

Although you can multi-select files in the **Data Source** view, when you click away from that view the highlighting applied to those files disappears. However, the files remain active and any subsequent operations are applied affect the files.

Table 9: Properties View: Definition Section

Property	Description			
	The unit system in which file is defined. Source points are interpreted in this Length Unit.			

Table 10: Properties View: Rigid Transformation Section

Property	Description	
Number Of Cop- ies	When set to zero (default), only the source mesh is transformed. If you specify a number of copies greater than zero, these will be in addition to the source mesh. For example, if you import a .cdb file with a single part and set Number Of Copies to 2, you will get 3 parts in Mechanical.	
Trans- form Original	This property is only available when Number Of Copies is set to 1 or greater. Select the checkbox if you want to apply the specified transformation to the source mesh.	
Origin X/Y/Z		

Property	Description
	These properties allow you to rotate the model about its origin in the XY, YZ, or ZX plane. If you specify any copies, the rotation will be applied relative to the previous copy (or source mesh in the case of the first copy).

Note

These transformations are applied in the following order:

- 1. Rotation about the Y Axis
- 2. Rotation about the X Axis
- 3. Rotation about the Z Axis
- 4. Translations
- e. Update property modifications (**Update Project**) and return to the **Project** tab.

Note

- You can modify any file in the Outline view by browsing to a new file using the browse option provided in the Location column.
- You can also delete files that you have selected (or multi-selected) by right-clicking one
 of the files in the **Outline** view and then choosing **Delete** from the context menu.
- 4. The **Setup** cell of the **External Model** system can be linked to a **Model** cell of a Mechanical system.

Transferring Data to Mechanical

The next step is to open your . cdb files in Mechanical.

- 1. To add a downstream Mechanical system:
 - Drag a valid analysis system from the **Toolbox** onto the project schematic.
 - Establish a link from the External Model [Setup] cell to the Mechanical system [Model] cell to complete
 the connection which will delete the Geometry cell. Multiple model cells in the Project Schematic can
 link to one analysis system. See Assembling Mechanical Models in the ANSYS Mechanical User's Guide
 for more details.
- 2. Right-click the **Model** cell and select **Properties** to verify the **Tolerance Angle** of the model.

The following properties are provided. Modify the properties as needed.

Tolerance Angle

The **Tolerance Angle** determines if adjacent elements are of the same face during the geometry creation process. The geometry creation process identifies groups of element facets on the exterior

of the mesh. These generated facets create geometric faces in Mechanical. Then skin detection algorithm scans the exterior element facets and groups them based on a tolerance angle. For example, two adjacent element facets are grouped into the same face if the angle between their normals is less than or equal to the given tolerance angle. Therefore, an angle tolerance of 180 degrees creates only a single face for the whole body while a tolerance of 1 degree creates an amount of geometric faces which approaches the number of element faces if any curvature is present.

Calculations to synthesize geometries using tolerance angles use the explicit method. This method is overridden if Nodal Components are processed.

The default **Tolerance Angle** is **45** degrees. This is the recommended setting.

Process Nodal Components

This option overrides **Tolerance Angle** during the geometry creation process if the .cdb file contains node-based components. And like **Tolerance Angle**, when node-based components span large portions of a model, clarity inaccuracies display in the graphical display of Mechanical.

Nodal Component Key

If .cdb files include nodal components, you can specify them using this property to further facilitate accurate geometries in Mechanical. Calculations to synthesize geometries using nodal components use the implicit method. This method overrides **Tolerance Angle** values if present.

Analysis Type

Define the .cdb file as **3D** (default) or **2D**.

3. Launch Mechanical.

Finite Element Modeler

Use the FE Modeler system to import a mesh and create a faceted (or NURBS) geometry to export to an analysis or geometry system. You can also create a parametric study within FE Modeler. Right click the **Model** cell and select **Edit** or **Import Mesh** to input an existing mesh file.

You can link many systems to an FE Modeler system by using the **Transfer Data From New** or **Transfer Data To New** context menu options. When transferring data from another system, you can transfer data from:

- Model Cell
- Setup Cell
- Mesh Cell

When transferring data to another system, you can transfer data to:

- Geometry Cell
- Engineering Data Cell
- Model Cell
- Mesh Cell

For more information on FE Modeler capabilities in ANSYS Workbench, refer to "FE Modeler System Usage in Workbench".

Fluent

Fluent allows for fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You specify the computational models, materials, boundary conditions, and solution parameters in Fluent, where the calculations are solved.

Use a Fluent component system to model incompressible and compressible fluid flow and heat transfer in complex geometries for your project. Within Fluent, a computational mesh is applied to a geometry, pertinent mathematical models are applied (e.g., low-speed, high-speed, laminar, turbulent, etc.), materials are chosen, boundary conditions are defined, and solution controls are specified that best represent the problem to be solved. Fluent solves the mathematical equations, and results of the simulation can be displayed in Fluent for further analysis (e.g. contours, vectors, etc.).

Drag the Fluent component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox. The Fluent component system is comprised of two cells: a **Setup** cell and a **Solution** cell. Double-click the **Setup** cell to open Fluent, where you can import a computational mesh, specify the computational models, materials, boundary conditions, and solution parameters, as well as perform the calculations. Alternatively, you can import a previously saved Fluent case file by right-clicking on the **Setup** cell and choosing **Import FLUENT Case**.

Fluent (with TGrid meshing)

Fluent (with TGrid meshing), also known as Fluent Meshing or Fluent in meshing mode, allows for meshing as well as fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You specify the mesh in Fluent in meshing mode, then proceed to set up the computational models, materials, boundary conditions, and solution parameters using Fluent in solution mode, where the calculations are solved.

Use a Fluent (with TGrid meshing) component system to model incompressible and compressible fluid flow and heat transfer in complex geometries for your project. Within Fluent in meshing mode, a computational mesh is imported and manipulated. Switch to Fluent in solution mode where pertinent mathematical models are applied (e.g., low-speed, high-speed, laminar, turbulent, etc.), materials are chosen, boundary conditions are defined, and solution controls are specified that best represent the problem to be solved. Fluent solves the mathematical equations, and results of the simulation can be displayed in Fluent for further analysis (e.g. contours, vectors, etc.).

Drag the Fluent (with TGrid meshing) component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox. The Fluent (with TGrid meshing) component system is comprised of three cells: a **Mesh** cell, a **Setup** cell , and a **Solution** cell. Double-click the **Mesh** cell to open Fluent in meshing mode, where you can import a computational mesh. Double-click the **Setup** cell to open Fluent in solution mode, where you can specify the computational models, materials, boundary conditions, and solution parameters, as well as perform the calculations. Alternatively, you can import a previously saved Fluent case file by right-clicking on the **Setup** cell and choosing **Import FLUENT Case**.

For detailed information on working with Fluent in meshing mode, see the Fluent Meshing User's Guide as well as the other on-line documentation available under the Help menu within Fluent in meshing mode. In addition, please see the Fluent in Workbench User's Guide.

For information on submitting Fluent jobs to Remote Solve Manager, see Submitting Fluent Jobs to RSM.

Geometry

Use the Geometry system to import a model. Right-mouse click the **Geometry** cell and select **New Geometry** or **Import Geometry**.

If you select **New Geometry**, the DesignModeler application opens. You can then build a model using the DesignModeler features. When you save the geometry in DesignModeler, the file becomes an .agdb file. If you select **Import Geometry**, you can browse to an existing geometry file.

You can also choose ANSYS SpaceClaim Direct Modeler via the **Tools** menu. For more information, see New Geometry in the DesignModeler section of the help.

You can connect other systems to a Geometry system by using the **Transfer Data From New** or **Transfer Data To New** context menu options. You can transfer data from the following types of systems:

- BladeGen: Connects the Blade Design cell to the Geometry cell.
- Finite Element Model: Connects the Model cell to the Geometry cell.

You can transfer data to the following types of systems:

- Mechanical APDL: Connects the Geometry cell to the Analysis cell. Transfer connection is via an .anf file.
- TurboGrid: Connects the **Geometry** cell to the **Turbo Mesh** cell.
- Vista TF: Connects the **Geometry** cell to the **Setup** cell. Transfer connection is via a .geo file.

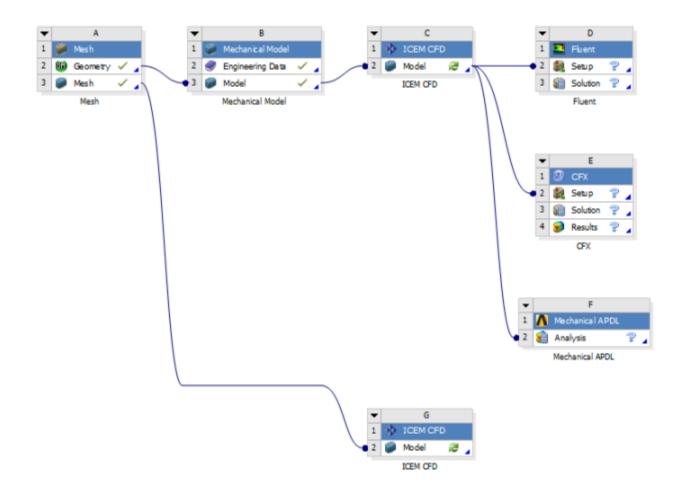
For more information on the geometry capabilities in ANSYS Workbench, refer to Project Schematic Operations in the DesignModeler User's Guide.

ANSYS ICEM CFD

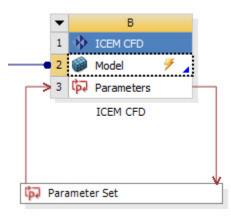
ANSYS ICEM CFD extends ANSYS meshing capabilities with robust and varied geometry import, the ability to efficiently mesh large or complex models with extended meshing controls, advanced interactive blocking tools for structured or unstructured mesh generation, extended mesh diagnostics, advanced interactive mesh editing and output to a wide variety of solver formats, including CFD, FEA and neutral formats.

ANSYS ICEM CFD can generate mesh from a range of sources including direct CAD, third party formats such as IGES or ACIS, faceted data, scan data or even combinations of CAD, facets and mesh. It includes a variety of "patch independent" meshing methods which are able to work with dirty cad and does not require that surfaces be formed into solids or that flow volumes be extracted. It does include a wide range of interactive geometry, blocking and mesh editing tools that can be used to generate advanced or high quality meshes for any application.

The data-integrated ICEM CFD component system, or "ANSYS ICEM CFD Add-in", enables you to launch ICEM CFD from ANSYS Workbench and use it to build a project, with the option of adding upstream data from Geometry, Mesh, Mechanical Model, or combined Geometry and Mesh system components. You can also use ICEM CFD to provide data to downstream component systems, such as ANSYS Fluent, ANSYS CFX, ANSYS Polyflow, Mechanical APDL, and FE Modeler.



Elements of the ICEM CFD Component



The ICEM CFD Component system contains the following cells:

• **ICEM CFD system header**. The System Header identifies the component type and provides access to Workbench context menu options. The ICEM CFD system header context menu options include:

- Refresh
- Update
- Duplicate
- Delete
- Rename
- Properties
- Add/Edit Note

These standard actions are described in System Header Context Menu Options (p. 270).

Note

If available, **Update** will use the **ICEM CFD Replay** file to update the ICEM CFD project.

Note

Scripts written in ICEM CFD may not be parametric with upstream or downstream projects. Care should be taken to write scripts whose functions do not exceed the capabilities of the upstream or downstream component systems.

- Model cell. The Model cell is associated with the ICEM CFD application. You can use the Model cell to
 modify some aspects of the project. You can also double-click the Model cell. to open the project in ICEM
 CFD. The Model cell context menu items include the following:
 - **Edit**: Opens the ICEM CFD application and loads an existing Geometry/ICEM CFD file.
 - Duplicate: Copies the entire geometry and mesh data, enabling you to edit the Model cell in the duplicate system to investigate an alternative modeling approach.
 - Transfer Data From New: Enables the transfer of data from upstream Geometry, Mesh, Mechanical Model, or combined Geometry and Mesh components.

Note

If the **Geometry** cell option **Use Associativity** is **On** (its default setting), Part Reference IDs are stored in a Workbench database and remain persistent in ICEM CFD even if the topology of the geometry is changed.

- Transfer Data to New: Enables the transfer of data from an ICEM CFD project to downstream data-in-tegrated system projects, such as:
 - → Fluent
 - \rightarrow CFX
 - → FE Modeler

- → Polyflow
- → Mechanical APDL
- Duplicate, Update, Refresh, Reset, Rename, Properties, and Add/Edit Note. These standard actions are described in System Header Context Menu Options (p. 270).

An additional property is **Create Subset(s) from Named Selection.** If this option is set, then overlapping named selections are transferred to multiple subsets instead of to exclusive parts. This enables the creation of geometry subsets instead of parts, allowing you to decide which part the geometry entity (point/curve/surface) should be associated to. When this option is disabled each geometry is assigned to only one part, thereby losing association with other Named Selections (which are mapped to parts).

 Parameters cell (optional). The Parameters cell enables you to see and edit Input and Output parameters for ICEM CFD.

Creating an ICEM CFD Component

You can create an ICEM CFD component system in Workbench using any of these methods:

- Double-click the ICEM CFD system template in the **Toolbox**.
- Drag-and-drop the ICEM CFD system template onto the Project Schematic.
- Right-click on a Geometry or Mesh project and select **Transfer Data to New> ICEM CFD**.
- Choose **File>Import** to import a standalone ICEM CFD project into Workbench. When the **Import** dialog opens, choose **ICEM CFD Project File (*.prj)** and navigate to the project you want to import.

Updating ICEM CFD Projects

Updating a project in Workbench brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.

Changes that require you to update your ICEM CFD project include changes to upstream data and changes you make in the ICEM CFD editor. When changes are made, the cell in the **Project Schematic** window indicates that an update is required. Note, however, that changes made in the ICEM CFD editor will not cause the system to go out of date until the project, tetin file (geometry), blocking file, and/or Replay file are saved.

Note

Named selections defined in Mesh systems are available only within the Mesh system. They are not available to downstream systems like ICEM CFD.

The actions taken by Workbench depend on whether the following conditions are met:

- · Blocking exists.
- A Replay file exists.
- · Blocking parameters are set.

• Other input parameters are set.

The following table describes the actions performed by ICEM CFD according to these conditions:

Table 11: Updating ICEM CFD Projects

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
No	No	No	No	 Runs tetra default meshing. Saves the unstructured mesh. Saves the project
Yes	No	No	No	 Runs hexa default meshing. Saves the unstructured mesh. Saves the project.
No	No	No	Yes	 Sets all input parameters. Runs tetra meshing. Runs prism meshing if any Part PART_NAME: Prism input parameters exist. (See Setting Parameters for Prism Meshing (p. 191).) Saves the unstructured mesh. Saves the project.
No	Yes	No	Yes	 Sets all input parameters. Runs the Replay file. Saves the unstructured mesh. Saves the project.

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
No	Yes	Yes	Yes	Sets all input para- meters except block- ing parameters.
				2. Runs the Replay file.
				3. If blocking now exists:
				a. Sets blocking in- put parameters.
				b. Runs hexa mesh- ing.
				c. Converts pre- mesh to unstruc- tured.
				d. Saves the unstruc- tured mesh
				4. Saves the project.
Yes	No	Yes	Yes	Sets all input parameters.
				2. Sets blocking input parameters.
				3. Runs hexa meshing.
				4. Converts pre-mesh to unstructured.
				5. Saves the unstructured mesh.
				6. Saves the project.
Yes	Yes	Yes	Yes	Sets all the input parameters except blocking.
				2. Runs the Replay file.
				3. If blocking still exists:
				a. Sets blocking in- put parameters.

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
				b. Runs hexa mesh- ing.
				c. Converts pre- mesh to unstruc- tured.
				d. Saves the unstructured mesh.
				4. Saves the project.

ICEM CFD saves the unstructured mesh and project only if the ICEM CFD GUI is closed and you update the project from Workbench. If the ICEM CFD GUI is open, you will need to manually save the unstructured mesh and project.

The order of operations is Meshing input parameters are set before the **Replay** file is run; Blocking Input parameters are set after a replay file is run, but only if blocking exists after the replay file is run.

If no Replay file exists, the default mesher is determined by the presence or absence of blocking: if no blocking exists in the project, **tetra** is the default. Conversely, if blocking exists in the project, **hexa** is the default mesher.

Updating a Project

To update a project and refresh upstream and downstream data, right-click on the **Model** cell and choose **Update**.

Interface Differences in the Data-Integrated ICEM CFD

The data-integrated ICEM CFD interface has been modified to provide additional functionality that enhances the integration of ICEM CFD and Workbench.

You can ensure that you are working within the data integrated environment by checking the **Message** window within ICEM CFD. The first line identifies that the application is integrated in the Workbench environment.

• One-click menus that enable you to control basic project and **Replay Recording** functions.

Note

If you create Workbench Input Parameters and want to step through the script line-by-line or using a line range, you must open the **Workbench Replay Control** dialog from this menu.

• The **Workbench Replay Control** dialog enables you to test and edit Replay scripts created within Workbench.

You can ensure that you are working within the data integrated environment by checking the **Message** window within IICEM CFD. The first line identifies that the application is integrated in the Workbench Framework.

ICEM CFD 14.5 in Workbench 2.0 Framework
Checked out ansystmd feature aienv (product ANSYS ICEM CFD) from server 1055@pghlnxlicense
Loading project settings file: ICM.prj...
Loading geometry file: "ICM.tin"
Current Coordinate system is global
Loading domain: "ICM.uns" ...
Loading family boco data from ICM.fbc
Current Coordinate system is global

One-Click Menus



The following one-click menu options are available in the **Toolbar** when you open ICEM CFD from Workbench:

- Save Project: Saves the entire project, including Workbench data.
- Refresh Project: Refreshes the upstream data in the ICEM CFD project.
- **Update Project**: Brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.
- Start Replay Recording: Begins recording the commands needed to generate a custom meshing process. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. See Replay Functionality in the ANSYS ICEM CFD User's Manual. After you click the Start Replay Recording Icon, the icon changes to the Stop Replay Recording icon. You can click this icon to stop recording.

You can also click on the arrow to choose **Pause Replay Recording**, **Run Replay File**, **Delete Replay File**, and **Replay Control**, which opens the **Workbench Replay Control** dialog.





Start Replay Recording



Pause Replay Recording



Stop Replay Recording



Run Replay Recording



Delete Replay File



Workbench Replay Control

Note

If you create Workbench Input Parameters and want to step line by line or using a line range through a replay file, you must use the **Replay Control** item from this menu to start the **Workbench Replay Control** dialog.

Output Mesh: You can choose to save the ICEM CFD mesh output to Fluent, CFX, or Polyflow projects.

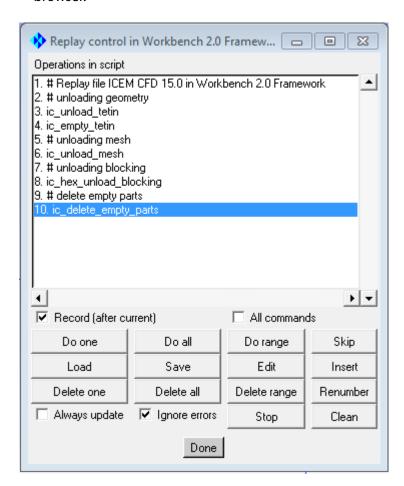


Workbench Replay Control Dialog

The **Replay Control** dialog helps you create, test, and edit script files by performing operations in ANSYS ICEM CFD and recording the equivalent Tcl/Tk commands in a Replay file. You can then use the dialog to step through and edit the script.

The **Workbench Replay Control** dialog works exactly the same as the standalone **Replay Control** dialog, with two key exceptions:

- The **Workbench Replay Control dialog** supports Workbench Input Parameters and allows you to step through them.
- The dialog automatically loads the current Replay script file (ICM.rpl) instead of opening a file browser.



For more information, see the Replay Scripts section in the ANSYS ICEM CFD Help Manual.

Setting Parameters

Setting Input parameters in Workbench enables you to pass parameters to ICEM CFD and other down-stream analysis tools. The interaction of parameters between applications provides you with greater flexibility and capabilities to run optimization and what-if scenarios. For more information about using parameters in Workbench, see Working with Parameters and Design Points (p. 93).

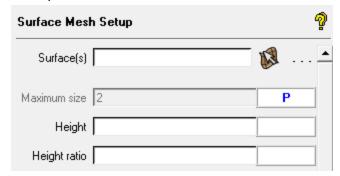
Parameters may be set globally or individually, with individual parameters taking precedence over global values.

The parameters you can set are:

- Input Parameters
- · Parameters for all existing curves, surfaces, or edges
- · Mesh parameters for parts
- User-defined Input Parameters
- Output parameters

Setting Input Parameters

Clicking the box to the right of certain **Meshing Input** parameters enables you to select whether the parameter is controlled from within ICEM CFD or from within Workbench. A "**P**" in the check box indicates that it has been selected as a Workbench Input parameter. If the check box is empty, you can control the input from within ICEM CFD.



You can set the following input parameters in Workbench:

- Global Mesh Size (See Global Mesh Size in the ANSYS ICEM CFD Help Manual.)
- **Shell Meshing** (See *Patch Dependent Options* in the *ANSYS ICEM CFD Help Manual*).
- **Volume Meshing** (See *Robust Octree* in the *ANSYS ICEM CFD Help Manual*).
- **Prism Meshing** (See *Global Prism Settings* in the *ANSYS ICEM CFD Help Manual*).
- Surface Mesh Setup (See Surface Mesh Setup in the ANSYS ICEM CFD Help Manual).
- **Curve Mesh Setup** (See *Curve Mesh Setup* in the ANSYS ICEM CFD Help Manual).
- Edge Params (See Edge Params in the ANSYS ICEM CFD Help Manual).

For **Surface Mesh Setup**, **Curve Mesh Setup**, and **Edge Params**, you can set parameters either on all existing surfaces or curves and edges at once, or for a single curve, surface, or edge.

Note

If you create Workbench Input Parameters and want to step line by line or using a line range through a replay file, you must use the **Workbench Replay Control** item from the **One-Click** menu to start the **Workbench Replay Control** dialog.

Setting Input Parameters

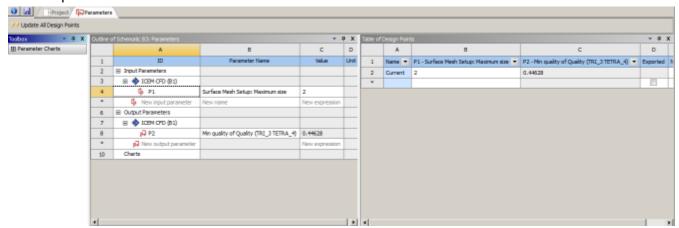
To set input parameters in Workbench:

- 1. Within ICEM CFD, choose any of the input parameters listed above.
- 2. Select the check box next to the parameter.
- 3. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter has been created for Workbench.

Note

You will not be able to edit the parameter within ICEM CFD unless you click the check box again and deselect the parameter.

- 4. In Workbench, double-click on the project's Parameters cell.
- 5. Edit the parameter values in the **Outline of Schematic: Parameters** window.



6. Click the **Project** tab to return to the **Project Schematic** window.

You can now update the project using the new parameter settings.

Setting the parameters for a single curve, surface, or edge

- 1. Within ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.
- 2. Click the **Select** button at the top of the **Parameters** window.

- 3. Click the **Left Mouse** button to select the curve, surface, or edge for which you want to set parameters.
- 4. Click the **Middle Mouse** button to complete the selection.

The surface, curve, or edge you selected are listed in the selection entry.



- 5. Select the check box next to the parameter you want as the input parameter.
- 6. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter for the single surface, curve or edge has been created for Workbench.

Setting the Parameters for All Existing Curves, Surfaces, or Edges

- 1. Within ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.
- 2. Leave the surface, curve or edge selection field empty (do not select any surface, curve or edge).
- 3. Select the check box next to the parameter you want as the input parameter
- 4. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter for all existing surfaces, curves or edges has been created for Workbench.

Setting Workbench Mesh Parameters for Parts

The **Part Mesh Setup** parameters enable you to specify the mesh parameters for different parts of a geometry. Part Mesh Setup is described in the *ANSYS ICEM CFD Help Manual*.

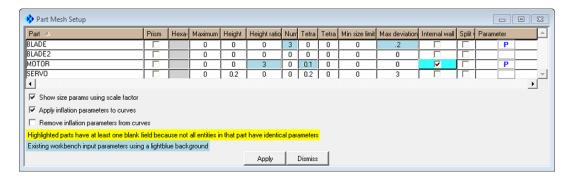
To create a Workbench mesh parameter for a part:

- 1. Click the **Mesh** tab.
- 2. Click the **Part Mesh Setup** icon.
- 3. In the **Part Mesh Setup** dialog, choose a part and assign non-zero values to one or more of its parameters.
- 4. Click in the **Parameter** column for the part.
- 5. A dialog asks you to confirm that you want to create a Workbench parameter for each value you've changed.

Note

If any Workbench parameters are already set for the part, a dialog will ask if you want to delete that Workbench parameter.

Parameters assigned as Workbench parameters are highlighted in blue.



To delete a parameter, click in the **Parameter** column. A dialog will ask you to confirm that you want to delete each individual Workbench parameter in the row.

As with ICEM CFD, the Apply inflation parameters to curves and Remove inflation parameters from curves options affect Workbench parameter behavior as well. The current value of Apply inflation parameters to curves is saved to the project file/.aienv_options file, so it is always available in GUI or Batch mode.

Setting Parameters for Prism Meshing

You can add prism meshing parameters using the Part Mesh Setup dialog:

- 1. Click the **Mesh** tab.
- 2. Click the Part Mesh Setup icon.
- 3. In the **Part Mesh Setup** dialog, choose a part and check the check box in the **Prism** column.
- 4. Click in the **Parameter** column for the part.
- 5. A dialog asks you to confirm that you want to create a Workbench parameter for **Part PARTNAME: Prism**.

Note

If any Workbench parameters are already set for the part, a dialog will ask if you want to delete that Workbench parameter.

Setting User-Defined Input Parameters

- 1. Within ICEM CFD, choose Settings>Workbench Parameters> Workbench Input Parameters (User-defined).
- 2. In the User-Defined Workbench Input Parameters window, check the Create User Defined Input Parameter check box.
- 3. Enter a value for the **Parameter name** (for example, MY_PARAMETER).
- 4. Enter a value for the **Parameter** (for example, 1.343). This value must not be empty.
- 5. Click **Apply** or **OK**.

You can edit this value in the **Outline of Schematic: Parameters** window.

Note

See the ANSYS ICEM CFD Programmer's Guide for information about using **User-Defined Parameters** with **Replay Scripting**.

Deleting User-defined Input Parameters

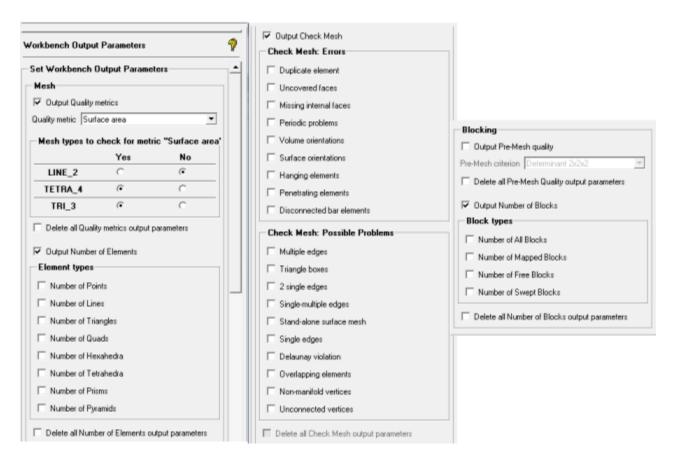
- 1. Within ICEM CFD, choose Settings>Workbench Parameters> Workbench Input Parameters (User-defined).
- 2. In the User-Defined Workbench Input Parameters window, check the Delete User Defined Input Parameter check box.
- 3. Use the drop-down menu to choose the name of the parameter you want to delete.
- 4. Click **Apply** or **OK**.

Setting Output Parameters

You can set Workbench Output parameters to:

- unstruct mesh and pre-mesh blocking quality metrics
- list the number of mesh errors and possible problems
- · list the number of element and block types created

You set the output parameters within the ICEM CFD application, then view them in either Workbench or ICEM CFD.



Setting Output Parameters

- 1. Within ICEM CFD, choose Settings>Workbench Parameters> Workbench Output Parameters.
- 2. Optionally, Click the **Output quality metrics** check box to select quality metrics.
 - a. Use the Quality Metrics drop-down menu to choose the metric you want to set.
 - b. Click the radio buttons to select the mesh types to check for the metric.
- 3. Optionally, click the **Output number of elements** check box.
 - a. Check the boxes next to the element types for which you want you want an output.
- 4. Optionally, click the **Output Check Mesh** check box.
 - a. Check the boxes next to the errors you want to have tallied in the output results. ICEM CFD will perform an analysis as part of the meshing process and list the results in the Output window and in the Workbench Parameters window.
- 5. Similarly, check the boxes for **Output Pre-Mesh Quality** and **Output Number of Blocks** to set the Blocking Output Parameters.

For more information about pre-mesh quality, see "Pre-Mesh Quality" in the ICEM CFD User's Guide.

For more information about Hexa Block types, see "Hexa Block Types" in the ICEM CFD User's Guide.

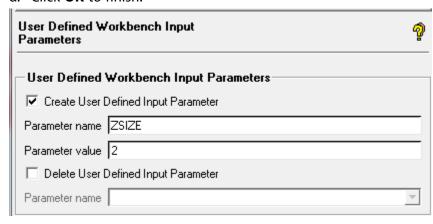
Deleting Output Parameters

- 1. Within ICEM CFD, choose Settings>Workbench Parameters> Workbench Output Parameters.
- 2. In the Workbench Output Parameters window, check the Delete all quality metrics output parameters check box and/or the Delete all number of elements output parameters check box.
- 3. Click Apply or OK.

User-Defined Parameters Example

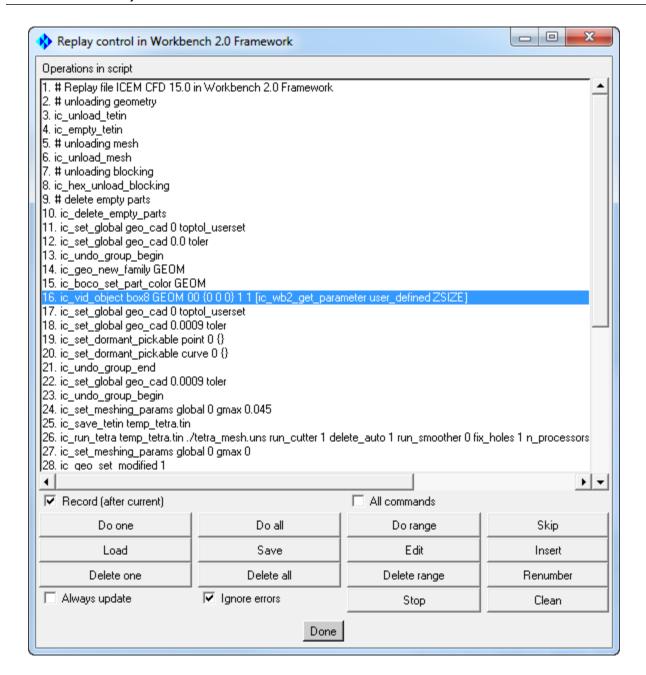
This example illustrates how you can use user-defined parameters to test different meshing scenarios for a simple box:

- In the Workbench Toolbox, double-click the ICEM CFD component to start the data-integrated ICEM CFD component system.
- 2. Double-click the **Model** cell to open ICEM CFD.
- 3. First, create an Input parameter, **ZSIZE**, that you can manipulate from Workbench.
 - a. Choose Settings>Workbench Parameters> Workbench Input Parameters (User-Defined).
 - b. Click Create User-Defined Input Parameter.
 - Name the Parameter ZSIZE and set the Parameter Value as 2
 - d. Click OK to finish.



- 4. Now create a box model upon which you will perform a meshing operation. Record the process so the operation can be performed again when you update it from Workbench.
 - a. Click the **Start Replay** button.
 - b. Click the **Geometry** tab and choose **Create/Modify Surface**.
 - c. Choose **Standard Shape** from the **Create/Modify Surface** window.
 - d. Choose **Box** and click **Apply**.
 - e. Click the **Mesh** tab and choose **Compute Mesh**.
 - f. Choose Volume Mesh and click Compute.

- g. Click **Yes** on the pop-up dialog to use autosizing for the mesh size.
- h. Click the **Stop Replay Recording** button.
- i. Choose **File>Save Project** to save the project.
- 5. Next, edit the script to use the input parameter.
 - a. Choose Replay Control from the One-Click menu.
 - b. Click the **Edit** button.
 - c. Edit Line 12 (or whichever line is appropriate), ic_vid_objectbox8 GEOM 00 {0 0 0} 1
 1 1, to ic_vid_object box8 GEOM 00 {0 0 0} 1 1 [ic_wb2_get_parameter
 user_defined ZSIZE]
 - d. Close the **Edit** window.
 - e. Click the **Save** button and save ICM.rpl.
 - f. Click the **Done** button. Do not close ICEM CFD.



- 6. Now you can change the Input parameter through Workbench.
 - a. In the Workbench **Project Schematic** window, double-click on the **Parameters** cell of the ICEM CFD component.
 - b. Change the value of **ZSIZE** to **5** and return to the **Project** tab.
 - c. Right click on the Model cell and choose Update.

You can watch the Replay script run using the new parameter in the ICEM CFD interface. Each time you change the **ZSIZE** parameter in Workbench, will be generated and meshed with the new **ZSIZE** parameter value.

Icepak

The Icepak application allows engineers to model electronic designs and perform heat transfer and fluid flow simulations. You can construct your model geometry or import model data from other CAD and CAE packages. Icepak then creates a mesh for your model geometry and passes the mesh and model definition to the solver for computational fluid dynamics simulation. The resulting data can then be postprocessed using Icepak and/or CFD-Post.

There are two ways to create a project in Icepak. In both cases you will first need to add an Icepak template to the project schematic. You can accomplish this by dragging the template from the Toolbox into the Project Schematic or double-click the template in the Toolbox. You will see the following in the project schematic:



Note

ANSYS Workbench provides visual indications of a cell's state at any given time with the use of icons on the right side of each cell. The descriptions of these icons are described in Understanding Cell States.

Note

A description of context menu options can be found in Context Menu Options

- 1. The first option for creating an Icepak project is described below:
 - a. Right click the **Setup** cell and select **Edit** or double click the **Setup** cell. This step will launch Icepak.
 - b. Create a geometry within the Icepak application. After creating a geometry, the system will show an incomplete cell state.



- indicates the system has not been solved.
- c. In Icepak, set up the problem and complete your analysis using Icepak's tools and features. After solving, the Icepak system will show an up to date cell state as shown below.



indicates that all necessary files are loaded and up to date.

- 2. The second option for creating an Icepak project is described below:
 - a. After adding an Icepak template, you can load an existing Icepak model by doing a right-mouse click on the **Setup** cell and selecting **Import Icepak Project**. You will find Icepak project files located in the system directory under *IPK*. All Icepak projects saved in ANSYS Workbench will follow this naming convention which is described in Project File Management. All other Icepak projects can be saved anywhere. Note, there are no special icons for Icepak projects to differentiate them from other files.

Note

You can only import one Icepak project per system.

You can also load an existing Icepak model by doing a right-mouse click on the **Setup** cell and selecting **Import Icepak Project From .tzr**. Click **Browse**. A file open dialog box will be displayed in which you can browse the directory structure and select the model to load. The system will show an incomplete cell state.



indicates the system has not been solved.

b. In Icepak, set up the problem and complete your analysis using Icepak's tools and features. After solving, the Icepak system will show an up to date cell state as shown below.





indicates that all necessary files are loaded and up to date.

Note

If you load an Icepak project that already has a solution, the **Solution** cell state will show an up to date cell state after the model is loaded.

3. You can also import geometry into Icepak from DesignModeler. Please see the DesignModeler to Icepak section for details on importing geometry from DesignModeler into Icepak or loading a geometry.

Workbench Menu Options Overview for Icepak Projects

Many Icepak options remain the same when running Icepak from the ANSYS Workbench framework. However, there are a few changes that should be noted.

Save project

Select Save project to save all changes to your project. The **Save** button in Workbench works the same way as **Save project** in Icepak, you can use either one. All Icepak projects saved in ANSYS Workbench will follow the directory structure described in Project File Management. See File Menu for a description of **Save** and **Save** as options in ANSYS Workbench.

Archive

Select **Archive** to generate a single archive file that contains all project files. This archive will include the project file and all files in the *project_name_files* directory, not just Icepak files. The archive will be saved as a zip file. Icepak users familiar with the Pack option in the standalone Icepak application will find this option works in a similar fashion except that all ANSYS Workbench files are archived. See File Menu for a complete description of the **Archive** option in ANSYS Workbench.

Note

Previously imported external files from a restored archive directory are treated as internal files if archived again.

Restore Archive

Select **Restore Archive** to restore a previously-generated archive file. After you select the project archive to be restored, you will be prompted for the name and location where the restored file(s) are to be located. After the archive is extracted, the project will open in ANSYS Workbench. See File Menu for a complete description of the **Restore Archive** option in ANSYS Workbench.

Note

You can also extract the archive manually by using an unzip utility, and then opening the .wbpj file.

Icepak Context Menu Options

When the Icepak system is active in the schematic, use the right mouse button to initiate the following **Setup** cell actions:

Edit

Launches Icepak. This is the default operation.

Import Icepak Project From .tzr

Brings up a **Browse** dialog box to add an input file, then launches Icepak and reads the input files. This is the default action. This option is available only when a project is empty.

Import Icepak Project

Brings up a **Browse For Folder** dialog box to add an input file, then launches Icepak and reads the input files. This is the default action. This option is available only when a project is empty.

Duplicate

Creates a duplicate of the Icepak system and any upstream data. If the duplicate operation is performed on a system containing a solution, the solution will not be transferred and the user will need to re-solve. The system cell state will indicate the need to re-solve.

Transfer Data From New

Creates a second, dependent (connected) system with the existing system.

Update

If data is changed or modified, you can perform an update. You have the option to incorporate the changes in the **Refresh Input Data** view and initiating the solver inside Icepak. See DesignModeler to Icepak for a further description of the update option and how the **Refresh Input Data** panel is used.

Note

You must open the Icepak editor before doing an update.

Refresh

If data is changed or modified, you can perform a refresh. You have the option to incorporate the changes in the **Refresh Input Data** panel. See DesignModeler to Icepak for a further description of the refresh option and how the **Refresh Input Data** panel is used.

Note

You must open the Icepak editor before doing a refresh.

Rename

Renames the system or cell.

Properties

Displays applicable cell properties in the **Properties** window.

Quick Help

Displays a quick help panel for the cell. Quick help provides a brief description of how to use the cell in its current state. You can also click on the blue triangle in the lower right corner of a cell to view quick help.

The **Solution** cell actions are similar to those of the **Setup** cell; however, there are additional actions and functions. These are described in the list below. Use the right mouse button to initiate the following **Solution** cell actions.

Set Case File...

Brings up an **Open** dialog box where you can specify an Icepak solution file to load. This option is used primarily to view multiple solutions for a project. If the user re-solves in Icepak, the solution case file will be overwritten with the latest solution.

Transfer Data to New

Creates a downstream system that can accept data from the selected cell. Only those systems that can provide valid data to the selected cell are shown. When you choose a system from the options shown here, that system will appear to the right of the currently-selected system, with all appropriate connections drawn.

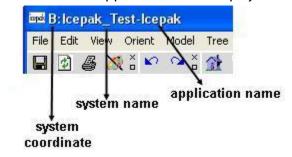
Update

If solution data is changed or modified, you can perform an update. You can bring in the latest solution case file into the solution cell for consumption.

System Names

You can enter an Icepak system name for your project as described in Naming Your Systems. In addition, you can use the **Rename** option to change the name of a system or cell. In the Icepak application you will find the system coordinate, system name and application name displayed in the top left corner.





Icepak Properties

Select or enable **Use Workbench Color Scheme** to keep the ANSYS Workbench background graphics colors and display defaults in Icepak. You can enable this option by performing a right mouse click on the **Setup** cell and selecting **Properties**. Enable the option by clicking the check box under the **Value** column.

DesignModeler to Icepak

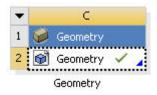
CAD models created or edited in DesignModeler can be imported into Icepak. Icepak instructs Design-Modeler to export the current DesignModeler geometry into Step file format so it is readable by Icepak. The procedure to transfer Geometry upstream to Icepak is described in this section.

1. Double click **Geometry** in the Toolbox under the **Component Systems** node or drag and drop the **Geometry** template into the Project Schematic.

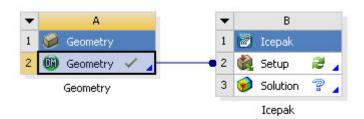


indicates that an input file is needed

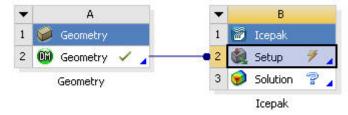
2. In the Project Schematic, double click the **Geometry** cell to launch the DesignModeler application or right mouse click to display the context menu. Select **New Geometry** to create a geometry in Design-Modeler. You can import any geometry or load an existing DesignModeler geometry by performing a right mouse click on the **Geometry** cell and selecting **Import Geometry** from the context menu.



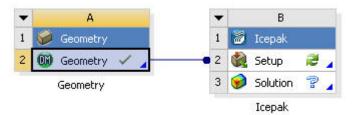
- indicates that all necessary files are loaded and up to date.
- 3. The connection to Icepak can be established one of the following ways:
 - Drag and drop an Icepak system on top of the **Geometry** cell to establish a connection.
 - Double click Icepak in the Toolbox under the Component Systems node to bring the Icepak system
 into the Project Schematic, click the Geometry cell and drag it to the Icepak Setup cell to establish
 a connection.
 - From the **Icepak** cell, perform a right mouse click and select **Transfer Data From New** in the context menu. You can then select a connection to DesignModeler.
 - From the **Geometry** cell, perform a right mouse click and select **Transfer Data to New** in the context menu. You can then select a connection to Icepak.



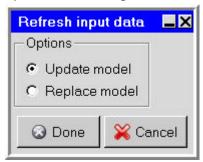
4. Double click on the Icepak cell to launch the Icepak application. If the geometry is different from the DesignModeler's native format (.agdb) then you will need to edit the geometry first before exporting into Icepak.



- 5. A new project will be created in the name of the project cell. DesignModeler geometry will be imported into Icepak as STEP geometry.
- 6. If the geometry is changed or modified, the Icepak **Setup** cell will need to be refreshed in order to bring in the new geometry.

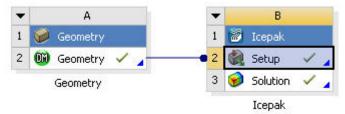


After you refresh the data, you will need to decide if you want to replace the entire model in Icepak or update individual geometries that have changed. You will be prompted with the following dialog box.

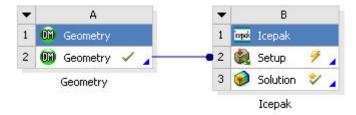


If you select **Update model**, the Icepak model will be updated to match the existing model in Design-Modeler which may result in creating objects that are not currently present in the Icepak model, the geometries of modified objects will be updated, and the thermal and material properties of the updated objects will be retained. Objects created separately in Icepak will not be updated. If you select **Replace model**, the entire Icepak model will be replaced with the latest input CAD data from DesignModeler. See the sample session in Chapter 2 of the Icepak documentation for an example on how to use Icepak. Refer to Understanding Cell States for a complete list of all cell states.

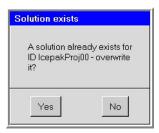
7. Perform an analysis on the model if you have not yet done so.



8. If the geometry has changed after you have solved, you can perform an update to bring in the new changes.



You can choose to not overwrite the solution via the dialog box shown below, and the update will be cancelled ($\frac{1}{2}$).



Note

See Workbench Menu Options Overview for Icepak Projects for information on how to save your project.

PostProcessing of Icepak Results

Icepak provides a two methods for examining the results of your simulation. You can postprocess results inside of Icepak or by using CFD-Post.

- 1. To postprocess results in Icepak, add an Icepak system to the project schematic and perform an analysis on the model. You can then create graphical displays and examine your results in Icepak. For details, see Examining the Results in the Icepak documentation.
- 2. Use one of the following ways to connect to CFD Post:
 - Drag and drop a **Results** cell on top of the Icepak **Solution** cell to establish a connection.
 - Double click Results in the Toolbox under the Component Systems node to bring the Results system
 into the Project schematic, click the Icepak Solution cell and drag it to the Results cell to establish
 a connection.
 - From the **Results** cell, perform a right mouse click and select **Transfer Data From New** in the context menu. You can then select a connection to Icepak.
 - From the Icepak cell, perform a right mouse click and select **Transfer Data To New** in the context menu. You can then select a connection to Results.



Icepak results can be postprocessed in CFD Post by double clicking on the Results cell. If you have
more than one solution or your solution has changed, you will need to update the Results cell. If
solution data is not transferred, then Results cannot be launched. Please see CFD-Post help for more
details.



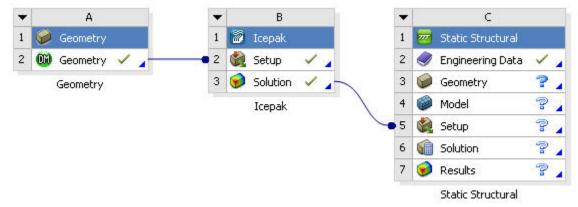
Thermal Results From Icepak to Mechanical

Icepak results can be transferred to the following analysis types within ANSYS Workbench: Static Structural, Steady-State Thermal, Transient Structural, or Transient Thermal. To add an analysis type, follow the procedure below:

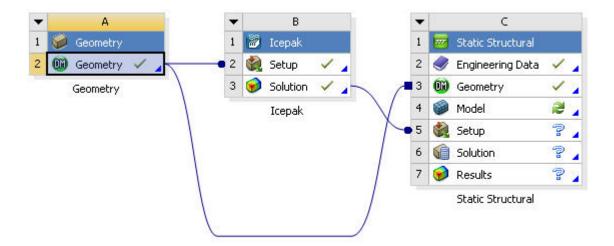
1. Add an Icepak system to the project schematic and perform an analysis on the model inside Icepak.



- 2. Use one of the following ways to connect to Mechanical.
 - Drag and drop an **Analysis** cell on top of the Icepak **Solution** cell to establish a connection.
 - Double click an analysis type in the Toolbox under the Analysis Systems node to bring the analysis into the Project schematic, click the Icepak Solution cell and drag it to the analysis **Setup** cell to establish a connection.
 - From the analysis **Setup** cell, perform a right mouse click and select **Transfer Data From New** in the context menu. You can then select a connection to Icepak.
 - From the Icepak cell, perform a right mouse click and select **Transfer Data To New** in the context menu. You can then select a connection to an analysis system.



3. Load geometry to satisfy the system.



- 4. Double click on the **Setup** cell to launch Mechanical.
- 5. While in Mechanical insert the appropriate loads based on the type of analysis. Please see Icepak to Mechanical Data Transfer for a detailed description of how to import an external load.

Design Explorer - Icepak Coupling in Workbench

The exploration of a given design can be performed by using optimization algorithms in ANSYS DesignXplorer. Parameters exposed from Icepak provide a method for solving an optimization and/or parameterization problem. To publish Icepak variables, follow the procedure below:

1. In Icepak, define input and output parameters. See Overview of Parameterization for a description on how to define parameters.

Note

The **Design variables** tab of the **Parameters and optimization** panel displays all the parameters names that are currently defined, along with their associated values.

- 2. In the **Setup** tab of the **Parameters and optimization** panel, select **Single trial (current values)** for **Trial type** if not already selected.
- 3. Click the **Publish to WB** button at the bottom of the **Parameters and optimization** panel to display the **Publish Variables** panel.

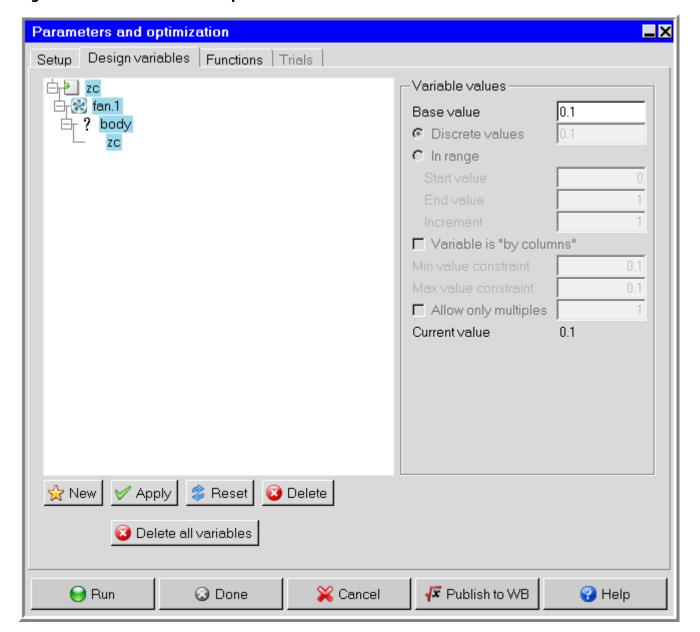


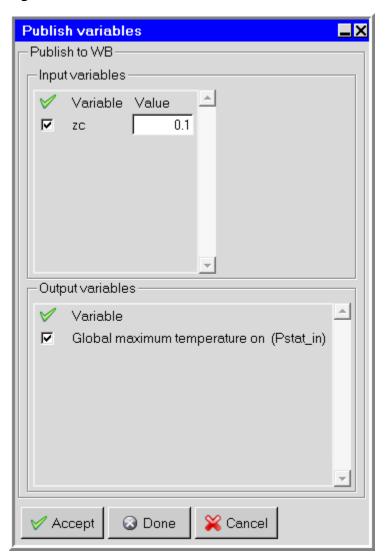
Figure 1: The Parameters and optimization Panel

• In the **Publish Variables** panel, select the input and output variables to publish to Workbench.

Note

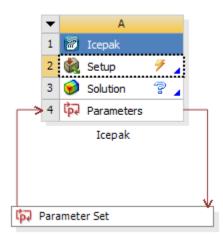
Variables can be selected independently. In the case of multiple variables, click the green check mark to toggle all variables on and off.

Figure 2: The Publish Variables Panel



Click Accept to publish variables to Workbench, click Done to publish variables and close the panel, or click Cancel to withdraw the request. ANSYS Workbench recognizes the parameters defined and exposes them in a parameter set bar that can be shared by multiple systems. Double-click the parameter bar or right-mouse click and select Edit from the context menu to access the Parameters tab. For information on defining parameters, refer to Working with Parameters and Design Points.

Figure 3: Icepak System with Parameters



Note

A design point is a set of input parameter values and corresponding output parameter values associated with an individual parameterized project definition. Design points can be created within the **Parameters** tab and allow you to perform what-if studies. For information on design points, refer to Design Points.

4. To create a new design point, enter the input parameter values that you want to use for that design point in the **Table of Design Points** in the row with an asterisk (*) in the first column. You can create several design points. Once you have finished specifying design points, you can right-click the row for one design point and select the **Update Selected Design Point** option from the context menu to compute the output parameters for that design point. Alternatively, you can select **Update All Design Points** from the Toolbar to update all of your design points in sequence.

Important

Only the data from the **design** point in the row labeled **Current** is saved with the project. If you want to post-process the results from a different design point in either ANSYS lcepak or ANSYS CFD-Post, click the box in the **Exported** column for that design point before you update that design point. Otherwise, the data for that design point is automatically deleted after the output parameters for that design point are updated. If you choose to export a design point, the data associated with that design point is exported to a new project. The new project is located in the same directory as the original project. The name of the project is the same as the name of the original project, except that it is appended with _dpn, where n is the row number that corresponds to the design point in the original project's **Table of Design Points**.

Important

Note that you cannot create, edit, delete, or rename parameters in Icepak if any iterations (or time-steps) have been performed. If you want to create, edit, delete, or rename

parameters in Icepak for a case with an existing solution, you must first initialize the solution.

5. Optimization of an Icepak system can be performed in ANSYS DesignXplorer. ANSYS DesignXplorer provides various optimization methods with parameters as its fundamental components. These parameters can come from any supported analysis system, such as Icepak, DesignModeler, and various CAD systems. Responses can be studied, quantified and graphed. For information on how to set up an ANSYS DesignXplorer analysis, see the Design Exploration User's Guide.

Ansoft - Icepak Coupling in Workbench

Icepak can be coupled with Ansoft applications within Workbench in order to perform a one-way electromagnetic-thermal interaction problem. Coupling between Ansoft and Icepak applications within Workbench can be used for simulating fluid flow around or inside electromechanical (EM) devices when the temperature of the device is influenced by electromagnetic losses.

Note

The Ansoft Maxwell, HFSS and Q3D Extractor applications are supported for one-way coupling between Ansoft and Icepak in Workbench.

The coupling involves solving an electromagnetic problem in the Ansoft application, and mapping the resulting volumetric heat loss and/or surface loss information into Icepak. Volumetric loss is mapped onto the solid cell zones as a heat source (load) at the cell centroids that is then added to the energy equation. Surface loss is applied to the adjacent cells of the solid zones at the surface that contribute to the source terms of these cells.

Note

Surface loss is highly concentrated near the surface of the solid zone, so it is recommended to have a fine layer of good quality hexahedral or prism mesh elements located where surface loss occurs.

You can analyze the results of volumetric or surface losses using the heat flow postprocessing variable under **Summary report**.

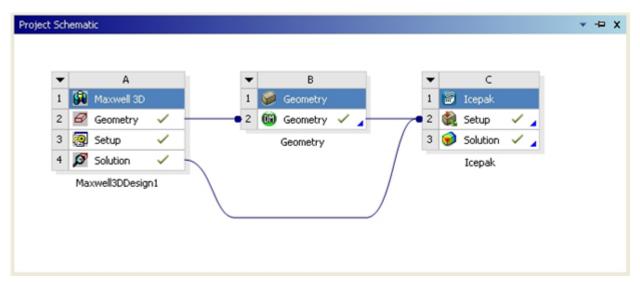
Note

When surface losses are enabled, the double precision solver is recommended.

The workflow for an Icepak- HFSS/Maxwell/Q3D Extractor analysis is as follows:

- The Ansoft to Icepak connection can happen one of the following ways:
 - Drag and drop a HFSS/Maxwell/Q3D Extractor system onto to the **Project Schematic**. Import/create
 the geometry in the Ansoft application. Setup the problem and solve to obtain a solution for transfer
 into an Icepak system. Connect the HFSS/Maxwell/Q3D Extractor **Solution** cell to the Icepak **Setup**cell.

Figure 4: Maxwell Transfer to Icepak



2. From the Icepak cell, you can perform a right mouse click and select **Transfer Data From New** in the context menu. You can then select a connection to HFSS/Maxwell/Q3D Extractor or from the geometry cell, perform a right-click and select **Transfer Data to New** in the context menu to connect to Icepak.

Figure 5: A Coupled HFSS-Icepak System



• Double click on the Icepak cell to launch the Icepak application. A new project will be created in the name of the project cell. If a DesignModeler system is used the geometry will be imported into Icepak automatically. Otherwise, the geometry needs to be created in Icepak using Icepak primitive objects.

Note

The geometry can be exported from the HFSS/Maxwell/Q3D Extractor cell to a **Geometry** cell and imported to the Icepak cell using DesignModeler Electronics.

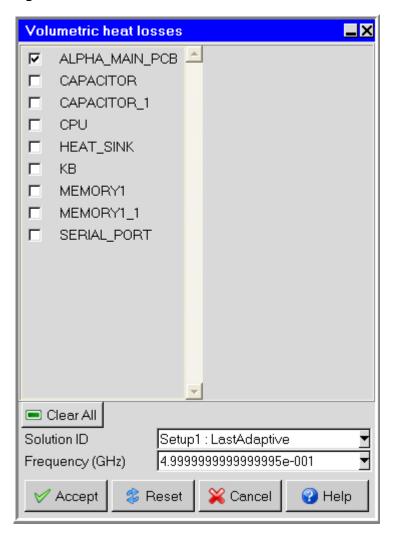
 In Icepak, go to the File menu and select EM Mapping and Volumetric heat losses or Surface heat losses.

The **Volumetric heat losses** panel contains the following inputs:

- A list of solid objects onto which the loss information can be mapped. For these objects, Icepak requests the heat source (loss) terms from the Ansoft application.
- Solution contains available solution sets. Since the HFSS/Maxwell/Q3D Extractor application may have multiple solutions, Icepak will request the generated heat source data for the selected solution.

 Frequency contains available frequencies. Icepak will request that the HFSS/Maxwell/Q3D Extractor application provide the heat source data for the selected frequency.

Figure 6: Volumetric heat losses



Click **Accept** to close the panel.

 In Icepak, click Start solution to solve the project. During solving, HFSS/Maxwell/Q3D Extractor will be launched in the background and the volumetric losses calculated and mapped onto the selected Icepak solid objects. This loss mapping from HFSS to Icepak is conservative. This is especially important for accurate temperature calculation.

Note

To calculate surface heat losses, select the **Surface heat losses** option and enter inputs as described above.

ANSYS Icepak - Workbench Integration Tutorial

This tutorial demonstrates how to create and solve an Icepak analysis in ANSYS Workbench. You will model a geometry using the direct CAD modeling feature in Icepak and create a non-conformal mesh

for the complex shapes. The project will also include postprocessing the results in CFD-Post and performing a static structural analysis.

In this tutorial you will learn how to:

- Create an Icepak Analysis in ANSYS Workbench.
- Solve a project and transfer to Mechanical for further analysis.
- · Postprocess results in CFD-Post.

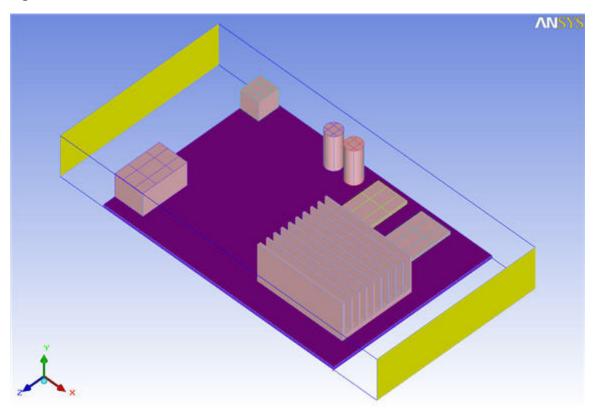
Prerequisites

This tutorial assumes that you have little experience with ANSYS Workbench and so each step will be explicitly described.

Problem Description

The graphics board contains a heat sink with extruded fins having aerofoil cross section, a PCB, capacitors, memory cards and ports. These objects are placed in a setup as shown in the figure below.

Figure 7: Problem Schematic

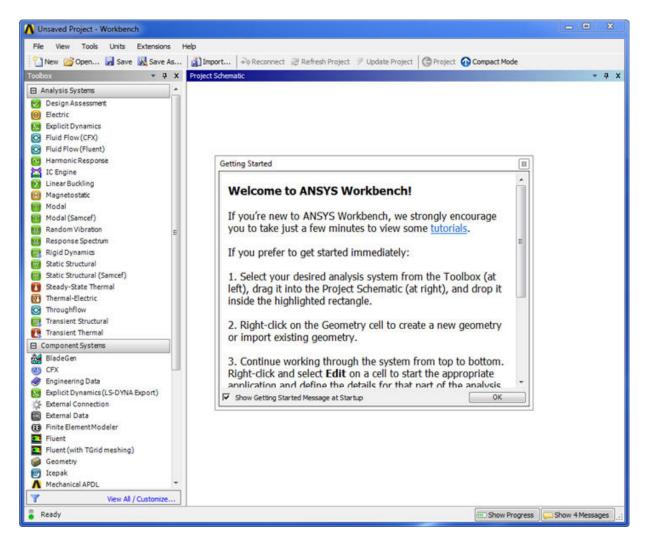


Step 1: Create a New Project

Start ANSYS Workbench.

Note

When ANSYS Workbench starts, the Toolbox and Project Schematic are displayed.



Step 2: Build the Model

Add a Geometry template by dragging the template from the Toolbar under the Component Systems
node into the Project Schematic. Perform a right mouse click on the Geometry cell and click Import
Geometry. Click Browse and select graphics_card_simple.stp to load the geometry.

Note

The graphics_card_simple.stp can be found at /ICEPAK_ROOT/tutorials/Workbench. You must replace ICEPAK_ROOT by the full path name of the directory where ANSYS Icepak is installed on your computer system.

Note

A green check mark in the **Geometry** cell indicates the geometry has been imported successfully.

2. Double click on the **Geometry** cell to open DesignModeler, you will need to edit the geometry first before exporting into ANSYS Icepak.

- 3. Select **Meter** as the desired length unit.
- 4. Click **Generate** to display the model.

Edit the geometry in DesignModeler using the **Electronics** option in the **Tools** menu. Select **Simplify** and choose the appropriate simplification level and select bodies. In this case, select all objects with level 2 **Simplification Type** and click **Generate**. Refer to the Design Modeler documentation for more detailed information on using the **Electronics** options.

Note

The Electronics menu is shown only if the DesignModeler option **Enable Electronics Options** is turned on.

- 5. Drag and drop an **Iceak** template into the Project Schematic on top of the **Geometry** cell (A2) to transfer the geometry into **Icepak**.
- 6. Right click on the **Setup** cell and select **Edit** to launch Icepak. The CAD model appears in the graphics display window and has been converted to **Icepak** objects. Click the isometric toolbar icon () to display the isometric view of the model.
- 7. Using the object edit panel, edit the properties for the objects. To edit the properties, perform a right mouse click on the object and select **Edit object** to display the object panel. Rename the objects using the object names as shown in The Final Model Display and click the **Properties** tab and enter the specifications shown below.

Table 12: Object Properties

Object	New name	Solid Material	Total Power
SERIAL_PORT	SERIAL_PORT	default	0.0 Watts
MEMORY1	MEMORY_1	Ceramic_material	5 Watts
MEMORY1.1	MEMORY_2	Ceramic_material	5 Watts
CAPACITOR	CAPACITOR_1	default	0.0 Watts
CAPACITOR.1	CAPACITOR_2	default	0.0 Watts
КВ	КВ	default	0.0 Watts
HEAT_SINK	HEAT_SINK	default	0.0 Watts
CPU	CPU	Ceramic_material	20 Watts

Object	New name	Solid Material	Total Power
ALHPA_MAIN_PCB	PCB	Custom- PCB solid_material	0.0 Watts
		Conductivity type- Ortho- tropic	
		X = 20, Y = 0.4, Z = 20	

Note

Edit the **Solid material** by selecting a material in the drop down list. To create a (Custom) material, select **Create material** in the drop down list and click the **Properties** tab in the **Materials** panel. Enter specifications.

8. Resize the default cabinet in the **Cabinet** panel.



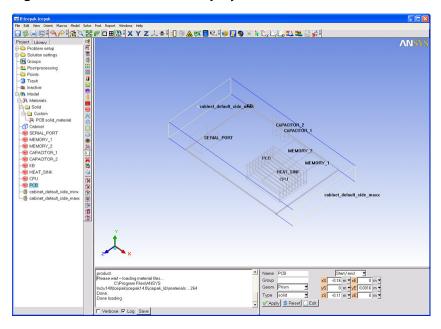
In the Cabinet panel, click the Geometry tab. Under Location, enter the following coordinates:

Table 13: Coordinates for the Cabinet

xS = -0.19 m	xE =0.03 m
yS = 0 m	yE = 0.02848 m
zS = -0.11 m	zE = 0 m

- 9. Edit the cabinet properties to specify Min x and Max x sides as openings. In the Properties tab of the Cabinet object panel, select Opening from the drop-down menu under Wall type for Min x and Max x. Select Edit to display the opening for the Max x object panel and specify the x velocity to be -2 m/s. Click Done to close the Openings and Cabinet panels.
- 10. The final model should correspond to the one shown below.

Figure 8: The Final Model Display



Step 3: Generate a Mesh

Note

For more information on how to refine a mesh locally, please refer to the ANSYS Icepak User's Guide.

• Click the assembly toolbar icon (to create an assembly. Add the heat sink and the CPU to the assembly and rename it CPU_assembly.

Note

To add objects to an assembly, select one or more objects in the Model manager window and drag them into the desired assembly node.

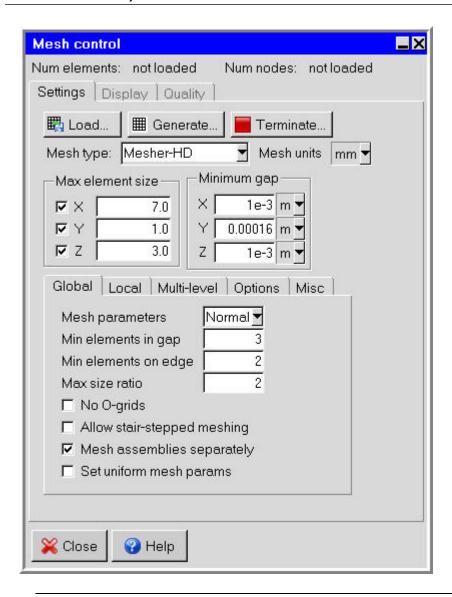
• Display the CPU_assembly object panel and click the **Meshing** tab. Enable the **Mesh separately** option and enter the following slack values. Click **Done** to close the panel.

Table 14: Slack values

Min $X = 0.005 \text{ m}$	Max $X = 0.005 \text{ m}$
Min Y = 0.0016 m	Max Y = 0 m
Min Z = 0.001 m	Max $Z = 0.005 \text{ m}$

• Specify the overall mesh controls, see the **Mesh control** panel below for input values.

Model → **Generate** mesh



Note

Change units to mm.

Step 4: Physical and Numerical Settings

• Problem setup → Basic parameters

Click on **Basic parameters** under **Problem setup** in the model tree. Under **General setup** make sure that both flow and the temperature fields are switched on. In addition, select **Turbulent** for the **Flow regime** and turn **Radiation Off**. Click **Accept** to close the panel.

• $\stackrel{ extstyle o}{=}$ Solution settings ightarrow Basic settings

In the model tree, go to **Solution settings**, then to **Basic settings** and **Advanced Settings** menus, and verify that the following values are set for each variable:

Basic settings No. of iterations = 100 Flow = 0.001 Energy = 1e-7 Advanced settings Pressure = 0.3 Momentum = 0.7

Step 5: Save the Model

Select the **File** menu and click **Save project**.

Note

The **Save As** panel will appear.

Specify the name "ice_wb" for your project and click **Save**.

Note

You can click the save icon (\Box) in the File commands toolbar.

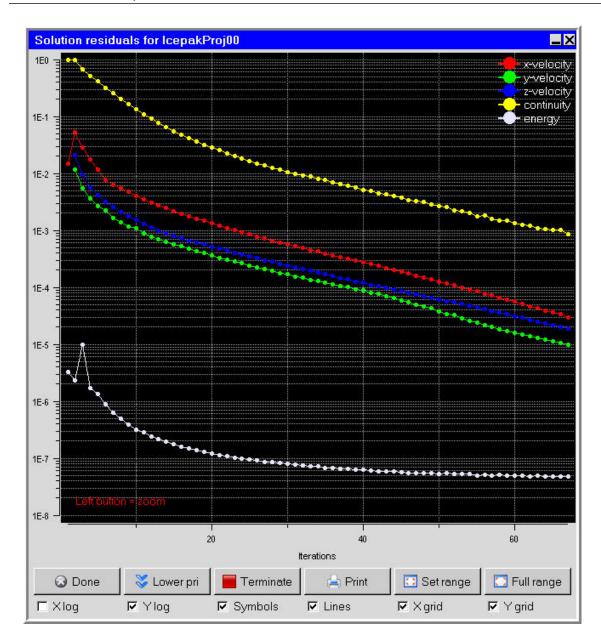
Step 6: Calculate a Solution

Solve → **Run solution**

- Select the **Solve** menu and click **Run Solution** to display the **Solve** panel.
- Keep the default settings in the Solve panel.
- · Click Start solution to start the solver.

Note

Icepak will begin to calculate a solution for the model, and a separate window will open where the solver will print the numerical values of the residuals. Icepak will also open the **Solution residuals** graphics display and control window, where it will display the convergence history for the calculation. Note that the actual values of the residuals may differ slightly on different machines, so your plot may not look exactly the same as the figure below.



• Click **Done** in the **Solution residuals** window to close it.

Step 7: Examine the Results with CFD-Post

Note

The postprocessing of results can be done within Icepak; however, you can examine results in CFD-Post. This section will describe how to transfer information to CFD-Post and use its postprocessing options.

After calculating a solution in Icepak, a green check mark will be displayed in the Icepak Solution cell
in the Project Schematic. The green check mark indicates that all data is up to date. Select Results under
the Component Systems node in the Toolbox. Drag the Results cell on top of the Icepak solution cell
(B3) to transfer the data.



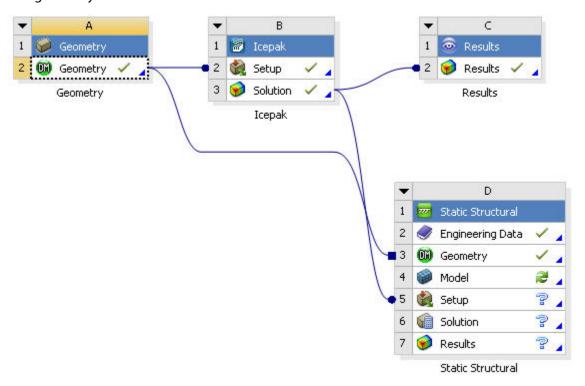
- 2. Double click the **Results** cell to launch CFD-Post. The model should appear in the display window.
- 3. To generate contours, please do the following:
 - Click the **Insert** menu and select **Contour** or click on the contour button to create a contour. Retain the name "Contour 1" and click **OK**.
 - In the **Details** section of the **Geometry** tab, select **All Domains** next to **Domains**.
 - Next to the **Locations** drop down box, click on the small box to display the **Locations Selector** dialog box. Highlight all CPU, PCB and HEAT_SINK objects and click **OK** to close the panel.
 - Next to Variable, select Temperature in the drop-down list.
 - Select **Apply** to display the contours.
- 4. To generate a 3D streamline, please do the following:
 - Click the **Insert** menu and select **streamline** or click on the streamline button to create the streamline. Retain the name "Streamline 1" and click **OK**.
 - In the **Details** section of the **Geometry** tab, select **3D Streamline** for **Type**.
 - Retain All Domains for Domains.
 - Select cabinet_default_side_maxx minx next to Start From.
 - · Retain Velocity for Variable.
 - Keep all other defaults the same.
 - Click **Apply** to display the streamline.
 - In addition you can animate the streamline. To animate the streamline, click on the animation button or select the **Tools** menu and click on **Animation**.

Step 8: Thermo- Mechanical Structural Analysis

Note

In addition to solving this problem in Icepak, you can also perform a static structural analysis.

- 1. Select **Static Structural** from the Toolbox and drag and drop this cell on top of the Icepak **Solution** cell (B3).
- 2. Right click on the **Geometry** cell (A2) and drop it on top of the **Static Structural Geometry** cell (D3). The geometry is now shared.



3. Double click on the **Model** cell (D4) to launch Mechanical.

12 - TE TE TE

- 4. Click on the **Imported Body Temperature** object. This object is found under the **Imported Load** (Solution) item.
- 5. In the **Details** section, ensure **Geometry Selection** is selected for the **Scoping Method**. Click the Box

Select button select it. Click **Apply** next to **Geometry**. Nine bodies should be selected.

- 6. Across from Icepak Body, select All.
- 7. Click Solve.

Step 9: Summary

In this tutorial, you imported CAD objects and set up a problem. You then created a non-conformal mesh using the hex-dominant mesher. This forced convection problem was solved for flow and heat transfer and the results were examined on contours and 3D streamlines in the model.

Mechanical APDL

You can use ANSYS Workbench to launch the Mechanical APDL application (formerly known as ANSYS), and thus to manage the various files often used and created by the Mechanical APDL application, especially when working with a linked analysis (e.g., thermal-stress, substructuring, submodeling, etc.). It is important that you understand the types of files that the Mechanical APDL application uses and generates, because the actions you take in ANSYS Workbench will act on these files. These files fall into three broad categories: input, reference, and output.

Input Files Files that are consumed directly by the Mechanical APDL application. Examples include:

- files consisting of Mechanical APDL commands, generated manually or by Mechanical APDL (log files) or by the Mechanical or Meshing applications.
- coded input files, such as .cdb files, generated by Mechanical APDL, FE Modeler, and third-party preprocessors
- Mechanical APDL geometry files (.anf), generated by Mechanical APDL or DesignModeler

Note

In some cases, the Mechanical APDL solver will overwrite one of its input files with its generated output (for example, the file .rst file from a Modal system in a Modal to Response Spectrum analysis linked to a Mechanical APDL component system). If this occurs, subsequent updates of the Mechanical APDL component system will fail. To copy the correct input from an upstream system, perform a Reset operation on the Mechanical APDL component system.

Referenced Files Files that are referenced by the execution of an input file. Examples include:

- · database files
- results files
- · command macro files
- superelement files
- solver files
- CAD geometry files

Output Files Files that are produced by all Mechanical APDL application runs. Primary output files include:

- results file (.rst, .rth, etc.)
- output file (.out) of the command echoes, solution information, and requested data listings
- log file (.log) of the commands issued to the Mechanical APDL application
- error file (.err) listing any warnings or errors encountered

Working with a Mechanical APDL Analysis

To add a Mechanical APDL analysis to your project, double-click the Mechanical APDL object or dragand-drop it from the Component Systems area of the Toolbox into the Project Schematic. A Mechanical APDL system appears in the Project Schematic.

You can easily connect other systems to a Mechanical APDL system by using the **Transfer Data From New** or **Transfer Data To New** context menu options. You can also drag systems from the toolbox, or manually create links between systems. When transferring data to a Mechanical APDL system from another system, you can transfer data from the following cell sources:

- **Geometry**: transfers just the geometry in the form of an .anf file. This option is only supported for geometry that is represented as DesignModeler geometry.
- **Model** (**Mesh** cell if a meshing system): transfers an input file containing only the mesh, contact, coordinate system, and named selections data
- **Setup** (Mechanical Systems): transfers an input file containing all data necessary to solve the analysis, including geometry, model, loads, materials, etc. Any supporting files needed to execute the input file will be transferred as well. Examples include pre-stress modal or random vibration.
- **Setup** (Finite Element Modeler): Transfers input file containing any finite data recognized inside Finite Element Modeler, such as mesh, materials, components, constraints, etc.
- **Solution** (Mechanical Systems): transfers the database file (.db) if it exists and result file only.

Note

For **Model**, **Setup** (Mechanical Systems), and **Solution** transfer cells, if you solve within Mechanical, you will still need to run an Update on the appropriate cell in the Mechanical system in order to the obtain the correct state on the schematic.

In most cases, **Model** and **Setup** components from the same Mechanical system should not be linked to one Mechanical APDL system. Doing so will cause the Mechanical system to provide two different (and possibly conflicting) input files to the Mechanical APDL system.

Important

The Mechanical APDL system consumes all input data without unit system knowledge. You must assure that all input data being used by the Mechanical APDL system is in a consistent unit system. Please see Solving Units for more information on unit system.

When transferring data from a Mechanical APDL system to another Mechanical APDL system, you can transfer four types of data:

- results: transfers all results files (including .rst, .rfl, .rth, etc.)
- database: transfers all database files (.db)
- solver: transfers all files in the system folder
- · CDB: transfers . cdb files

You can also transfer data to a new Finite Element Model system, which uses the .cdb file(s).

These files are simply copied to the new system if they exist; ANSYS Workbench does not generate the files. Before transferring data to a new system, be sure that you have an input file that generates the necessary files from the existing Mechanical APDL system.

When you transfer data to or from another system, right-mouse click the link connecting the systems and select **Properties**. The Properties window will open, detailing the nature of the transfer (such as Transfer CDB File).

Note on connecting to Mechanical systems Named Selections and Coordinate Systems that are added to a solved Mechanical system will not be immediately reflected in downstream Mechanical APDL systems. They will be available in future solution attempts.

To open the Components workspace to select Mechanical APDL parameters, double-click the **Analysis** cell or right mouse-click and select **Edit**. From the Components workspace, you can select Mechanical APDL parameters or specify setup properties (such as command line options, memory settings, number of processors, etc.).

When you add an input file via the context menu, ANSYS Workbench automatically searches the file for potential parameters (*SET, *GET, = assignments, etc.). Those parameters are then displayed in the Properties view when that input file is selected in the Outline view. To use one of those parameters, check that parameter's check box in the Property view and indicate whether it should be used as an input or an output parameter. Input parameters are sent to Mechanical APDL with the value specified upon Update. After the Update, ANSYS Workbench retrieves the output values from Mechanical APDL and sets those values in ANSYS Workbench.

Note

The presence of a **/EXIT** command in the input file causes state and parameters to malfunction. Make sure you remove this command before adding the input file.

To launch the Mechanical APDL application interactively, right mouse-click and select **Edit in Mechanical APDL** or **Open in Mechanical APDL** as explained below in Mechanical APDL Context Menu Options (p. 226).

To launch the Mechanical APDL application with input and reference files specified, right mouse-click the **Analysis** cell and select **Add Input File** or **Add Reference File**. Then select **Edit in Mechanical APDL**. The Mechanical APDL application will launch in interactive mode, and the input file(s) specified will be piped to the Mechanical APDL application, and processed in the order listed. After all of these files are processed, the Mechanical APDL application remains active and you can continue your analysis using the standard Mechanical APDL application interface. Any action you take in the Mechanical APDL application will not be reflected in ANSYS Workbench state indicators or parameters.

Be aware that any time you launch the Mechanical APDL application, ANSYS Workbench does not log or record the actions that occur in the Mechanical APDL application. If you make changes in the Mechanical APDL application, be sure that the changes are reflected appropriately in the input files. To maintain connectivity (such as to read output parameters), use the Update capability, either at the project level or at the appropriate system/cell level.

To save Mechanical APDL changes from an open session, you must include a **SAVE** command in one of your input files. The ANSYS Workbench save capability does not invoke the Mechanical APDL **SAVE** command.

When you add an input file, you will see the files listed in the files detail view. Files will be processed in the order shown. You can change the order in which the files are processed by dragging the files into the proper order. To delete files, right-mouse click the file to be deleted and select **Delete**.

To stop a Mechanical APDL batch run, view the Progress window. Click the Stop button on the Progress cell.

Mechanical APDL Context Menu Options

When the Mechanical APDL system is active in the schematic, use the right mouse button to initiate the following **Analysis** cell actions.

Edit

Opens the Components workspace, where you can specify Mechanical APDL parameters and setup properties. This is the default action.

Edit in Mechanical APDL

Launches the Mechanical APDL application interactively and reads the input files. If the state is currently up to date, ANSYS Workbench sets the state to Update Required at this time, even if you do not make any changes in the Mechanical APDL application.

Open in Mechanical APDL

Launches the Mechanical APDL application interactively without reading any input files. Any action you take in the Mechanical APDL application will not be reflected in ANSYS Workbench state indicators.

Add Input File

Displays a **Browse...** dialog box to add an input file. When you add an input file, the file is immediately copied into the project directory. To make changes to this file, change the file in the project directory, not the original file. If you have a large input file and have disk space concerns, keep the file in the directory of your choice and use a separate input file to reference it (via the /INPUT command).

Add Referenced File

Displays a **Browse...** dialog box to add a referenced file. When you add a referenced file, the file is immediately copied into the project directory. To make changes to this file, change the file in the project directory, not the original file. If you have a large reference file and have disk space concerns, keep the file in the directory of your choice and reference it manually.

Track Solution

During an Update, this option launches the Results Tracking tool, allowing you to monitor diagnostics results of interest in real time during the solution. For more information, see the **NLHIST** command.

Update

Update runs the Mechanical APDL application in batch mode, processing all input files in the order listed. If you make changes in the Mechanical APDL application, be sure that the changes are reflected appropriately in the input files before running an Update. Otherwise, an Update could potentially overwrite the work you've done in the Mechanical APDL application.

Note

An Update will launch the Mechanical APDL application in batch mode, using all input and referenced files in the order shown in the Outline pane. After all files are processed, the Mechanical APDL application exits. Updating will capture any output parameters

generated in the Mechanical APDL application and allow you to continue working in ANSYS Workbench.

Refresh

Copies the latest transfer files into the project directory. Input and referenced files are not re-copied from their original locations. If you change an upstream system after you make changes to the Mechanical APDL application, a refresh could potentially overwrite your Mechanical APDL application changes. Be sure that any changes you do in the Mechanical APDL application are reflected appropriately in the input files before running a Refresh. Only changes that occur within the schematic are captured with a Refresh Required state; ANSYS Workbench will not indicate Refresh Required for changes made directly to a file (such as manually editing an input file).

Clear Generated Data

Deletes all files on disk in the system directory except input or reference files. It will not affect any input or reference files.

Rename, Reset

Standard actions as described in Common Context Menu Options (p. 265).

Properties

Launches the Properties window, where you can define graphics settings, command line options, database and tab memory, and other settings. Be aware when selecting graphics settings that some options are potentially platform-specific and must be changed when switching platforms before running the project with Mechanical APDL in interactive mode.

Note on the solver input file generated from the Setup cell of a Mechanical APDL system The solver input file transferred from the Setup cell contains all the commands needed to execute a complete run, including any SOLVE commands that are necessary. However, the input also contains a conditional /EOF statement to halt reading of the file and thus not execute the solve. This conditional statement will be executed when the Mechanical APDL application is invoked from the Edit in Mechanical APDL context menu option, thus running the analysis to the point just prior to the SOLVE command. If a different behavior is desired, you can edit the input file in the Mechanical APDL system folder to obtain a different behavior or to add an additional input file containing the SOLVE command.

Mechanical Model

A Mechanical Model system consists of **Engineering Data**, **Geometry**, and **Model** cells. In the Mechanical application, this system corresponds to that of a Model-only system. You can use this type of system to create an analysis using a single model and multiple system analysis branches. You can also create other Mechanical Model systems that share data at any cell level (**Engineering Data**, **Geometry**, or **Model**). Mechanical Model systems are unfiltered (physics and solver).

The Mechanical Model system is also created when you resume a legacy database that does not have an analysis environment already defined. A Mechanical Model system may also be used as a system replacement for a Mesh system.

To create a new Mechanical Model system:

- 1. Choose Mechanical Model from the Component Systems section of the Toolbox. Double-click or drag the Mechanical Model system onto the Project Schematic.
- 2. Create or attach a geometry using the **Geometry** cell context menu.

- 3. Edit the model if necessary. Right-mouse click the **Model** cell and choose **Edit**.
- 4. Add a connected system either by dragging a valid analysis system from the Toolbox and dropping it on the appropriate target location, or right-mouse click the **Geometry** or **Model** cell and choose **Transfer Data To New**.
- 5. To create multiple system analysis branches, repeat step 4 with other analysis systems.

To resume an existing legacy database that contains no physics environment:

- 1. Choose **File > Import**. Browse to and select the legacy database and click **Open**.
- A Mechanical Model system appears in the Project Schematic, with the legacy database loaded. Doubleclick the **Model** cell, or choose **Edit** from the **Model** cell context menu to open the Mechanical application.
- 3. In the Mechanical application, proceed with any necessary Model updates.
- 4. You can then add any valid analysis system by dragging a template from the Toolbox to the Project Schematic. Choose the appropriate drop location that shares the desired cells with the Mechanical Model system.
- 5. Continue with the analysis in the analysis system as you normally would.

To replace an existing Mesh system with a Mechanical Model system (or vice-versa), select **Replace** with and choose **Mechanical Model** or **Mesh** (respectively) from the header context menu. When a Mesh system is replaced with a Mechanical Model system, the Mechanical Model system can then be shared with any analysis system as described above.

The units setting specified in an existing system is not maintained in the replacement system. In the replacement system, you must select the units setting that you want to use.

Model-to-Model Connections

Multiple Mechanical Model component systems can be merged together by creating a connection between the **Model** cells of each component system. This allows you to build up more complicated models from smaller, more simple models. This behaves in the same way as Mesh-to-Mesh Connections (p. 230). More detailed information can be found in Assembling Mechanical Models in the ANSYS Mechanical User's Guide

Mesh

You can use the Mesh component system to create and/or open geometry or mesh files. The Mesh component system contains a **Mesh** system header and two cells.

You can create a Mesh component system using any of these methods:

- Double-click the Mesh system template in the Toolbox.
- Drag-and-drop the Mesh system template onto the Project Schematic.
- Drag-and-drop a .meshdat or .cmdb file from Windows Explorer onto the Project Schematic.
- Choose File>Import or click the Import button from ANSYS Workbench and select a file of type
 .meshdat or .cmdb.

Mesh Context Menu Options

The Mesh component system contains a **Mesh** system header:

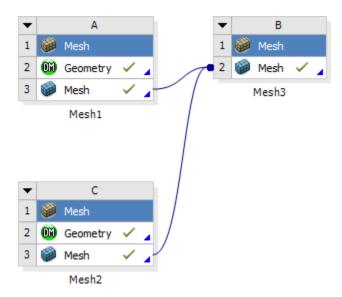
- 1. **Mesh**: Mesh system header context menu options include the following:
 - · Replace With> Mechanical Model
 - Refresh, Update, Duplicate, Delete, Rename, Properties: Standard actions as described in System Header Context Menu Options (p. 270).

The Mesh component system contains two cells:

- 1. **Geometry**: **Geometry** cell context menu options include the following: **New Geometry**, **Import Geometry**, **Duplicate**, **Transfer Data From New**, **Transfer Data To New**, **Update**, **Refresh**, **Reset**, **Rename**, **Properties**. For details see Geometry (p. 253).
- 2. Mesh: Model cell context menu items include the following:
 - Edit: Opens the Meshing application and loads an existing geometry/mesh file.
 - Import Mesh File: Allows you to import read-only meshes for downstream application use. You may browse to local files using the **Browse** option or to files stored on an EKM repository using the **Browse** from Repository option. For more information on **Browse from Repository**, see Importing Repository Files (p. 131).
 - **Duplicate**: Duplicates the Mesh system. The **Geometry** cell is shared, and all data associated with the **Mesh** cell is copied to the second system.
 - **Transfer Data To New**: Enables the transfer of a mesh from the Mesh system into a downstream system as follows:
 - Autodyn: Inserts downstream Autodyn system and generates a data transfer connection (.cmdb file) from the Mesh cell to the Setup cell of the Autodyn system.
 - CFX: Inserts downstream CFX system and generates a data transfer connection (.cmdb file) from the Mesh cell to the Setup cell of the CFX system.
 - FE Modeler: Inserts downstream FE Modeler system and generates a data transfer connection (.cmdb file) from the Mesh cell to the Model cell of the FE Modelerr system.
 - Fluent: Inserts a downstream Fluent system and generates a data transfer connection (.msh file) from the Mesh cell to the Setup cell of the Fluent system.
 - Mechanical APDL: Inserts a downstream Mechanical APDL system and generates a data transfer connection (.inp file) from the Mesh cell to the Analysis cell of the Mechanical APDL system.
 - Polyflow: Inserts a downstream Polyflow system and generates a data transfer connection (.poly file) from the Mesh cell to the Setup cell of the Polyflow system.
 - Update, Refresh, Clear Generated Data, Reset, Rename, Properties: Standard actions as described in Common Context Menu Options (p. 265).

Mesh-to-Mesh Connections

Multiple Mesh component systems can be merged together by creating a connection between the **Mesh** cells of each component system. This allows you to build up more complicated meshes from smaller, more simple meshes.



For every upstream **Mesh** cell connected to a downstream **Mesh** cell, a new set of properties called **Rigid Transformation for [Mesh component name]** will appear in the Properties view of the downstream **Mesh** cell. In the above example, with two upstream Mesh components connected to single, downstream Mesh component, the Properties view of the downstream Mesh (cell B3) would have two new groups of properties.

13	■ General Mesh Assembly Properti	es	
14	Length Unit	m 💌	
15	■ Rigid Transformation for Mesh1	(Component ID: Mesh)	
16	Number of Copies	1	
17	Transform the Original		
18	Origin X	3	m 🔻
19	Origin Y	0	m 🔻
20	Origin Z	0	m 🔻
21	Theta XY	0	radian 💌
22	Theta YZ	0	radian 💌
23	Theta ZX	0	radian 💌
24	■ Rigid Transformation for Mesh2	(Component ID: Mesh 2)	
25	Number of Copies	1	
26	Transform the Original		
27	Origin X	0	m 🔻
28	Origin Y	0	m 🔻
29	Origin Z	0	m 🔻
30	Theta XY	0	radian 💌
31	Theta YZ	0	radian 💌
32	Theta ZX	90	radian 💌

Table 15: Properties View: General Properties

Property	Description
	This property sets the units for the downstream mesh assembly. After the meshes are initially assembled, this property becomes read-only. Afterwords, you will have to reset the downstream mesh cell in order to change the Length Unit .

Table 16: Properties View: Rigid Transformation Section

Property	Description
Number Of Copies	When set to zero (default), only the source mesh is transformed. If you specify a number of copies greater than zero, these will be in addition to the source mesh. For example, if you import a .cdb file with a single part and set Number Of Copies to 2, you will get 3 parts in Mechanical.
Transform Original	This property is only available when Number Of Copies is set to 1 or greater. Select the checkbox if you want to apply the specified transformation to the source mesh.
Origin X/Y/Z	These properties allow you to translate the origin of the model along the X, Y, or Z axis. If you specify any copies, the translation will be applied relative to the previous copy (or source mesh in the case of the first copy).
Theta XY/YZ/ZX	These properties allow you to rotate the model about its origin in the XY, YZ, or ZX plane. If you specify any copies, the rotation will be applied relative to the previous copy (or source mesh in the case of the first copy).

Any change in these properties will put the downstream mesh in a state of refresh required.

Microsoft Office Excel

Microsoft Office Excel 2007 or 2010 can be used as a calculator in ANSYS Workbench via the "Microsoft Office Excel" system template found in the Component Systems toolbox. This feature exposes Excel ranges as input and output parameters to ANSYS Workbench which can be used to create Design Points and Design Exploration studies.

Note: The Excel add-in is included with ANSYS Workbench. It does not require DesignXplorer or any other specific ANSYS product, but it must connect to a locally installed seat of Microsoft Excel (purchased separately).

Using Excel with ANSYS Workbench Projects topics:

Preparing the Excel file
Set Up the Excel Calculator
Support of Units
File management and modification of the worksheet
Limitations
Troubleshooting

Preparing the Excel file

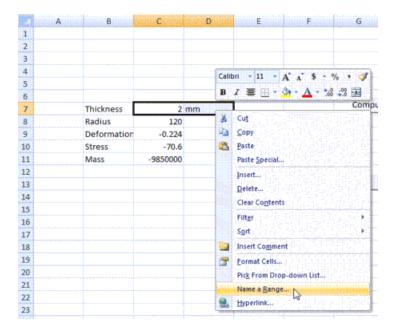
To be exposed as parameters in Workbench, the ranges must be named in Excel. The names are filtered during the addition of the Excel file to the project; only the ranges matching the prefix defined by the **Named Range Key** property in the **Properties** view of the **Setup** object are made visible in ANSYS Workbench.

Note

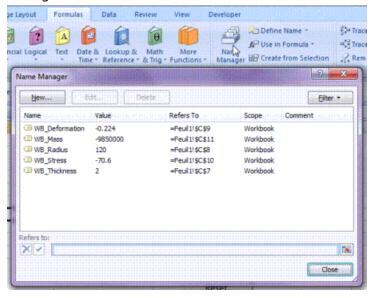
By default, no filtering prefix is defined at either the Workbench or the project level. You set a filter by either of the following methods:

- Set a default prefix that will be used for all new projects (see Microsoft Office Excel Options).
- Set a prefix at the project level by entering it in the **Named Range Key** property in the **Properties** view of the Microsoft Office Excel **Setup** object.

To name a range in Excel, open the file in the Microsoft Office Excel application, select a cell, right-click and select the **Name a Range** menu entry, enter the name and validate.



You can review and modify all the defined names in the Excel application using Formulas > Name Manager.



A named range can contain several cells. Workbench expects the named ranges to contain either one or two cells, the first cell being reserved for the value and the second cell being reserved for the unit string, if any. If a named range contains more than two cells, it is ignored.

Note

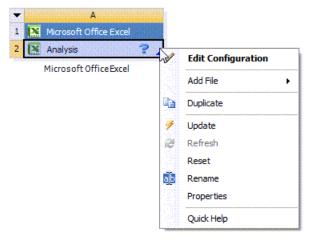
If you added the file to the Excel system in Workbench before naming the ranges, or if you wish to edit the names of the ranges after the addition of the file, you can right-click on the **Analysis** component or the file node in the **Outline** and select the **Open file in Excel** operation. This opens the Excel file used by Workbench in Excel.

All changes performed in Excel impact the state of the data in Workbench and invalidate the results; be sure to save your changes and Reload the file afterwards (when editing the **Analysis** component, right-click on the file node and select **Reload**).

Set Up the Excel Calculator

Adding a File

Drag and drop a "Microsoft Office Excel" system template on to the Project Schematic. Right-click on the Analysis system, select **Add File** and browse to the Excel file that you have prepared.



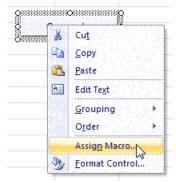
Then right-click on the Analysis system and select **Edit Configuration** to review the list of named ranges retrieved from the Excel file. For each listed named range, check the Input or Output column in order to publish the range as an input parameter or an output parameter in the Workbench project. Return to the Project Schematic; the Microsoft Office Excel is linked to the Parameter Set bar.

Macro Property Usage

If the calculation in Excel requires the execution of a macro, select the added file in the outline and check its "Use a Macro" property in the Properties View. Then enter the name of the macro in the "Macro Name" property.

Note

If a button is used to start the calculation and you do not know the name of the macro associated with it, right-click the button and select **Assign Macro** to discover the name to use.

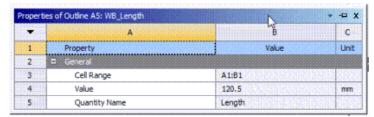


The project is ready to create Design Points and Design Exploration studies.

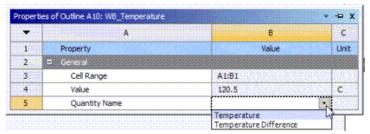
Support of Units

Units are handled by the Excel system. A valid Workbench unit string must be used and included in the named range.

For instance, if an input parameter is a length in millimeters, you can name "WB_Length" the range A1:B1 where the cell A1 contains the length value (i.e. "120.5") and the cell B1 contains the unit string (i.e "mm"). When selecting the "WB_Length" range in the Outline view, the Properties view looks like below. In this case, the Quantity Name is automatically identified as a Length.



If there are several possible Quantity Names for the same unit string, the user has to select the Quantity Name in the properties of the range for the unit conversion to be performed as expected. For instance, if A1:B1 was actually a temperature in degree Celsius, the Quantity Name could be a Temperature or a Temperature Difference.



For detailed information on working with units in ANSYS Workbench, see Unit Systems in the Working with Units section of the ANSYS Workbench help.

File management and modification of the worksheet

When the Excel file is added to the Analysis component, it is copied inside the Workbench project files. So any modification made to the original file is not seen by Workbench except if you delete and add the file again.

To modify the file copied in the Workbench project files, right-click on the Analysis component, or the file node in the Outline view, and select the **Open file in Excel** operation. Once modifications are done, save the file. The state of the file in Workbench changes to Refresh Required, which indicates that data are not synchronized anymore. Results such as Design Points and Design Exploration systems in the Workbench project are outdated. Refresh the project to synchronize all the pieces of the project.

If a change in the Excel file was not detected by Workbench, it is possible to force a reload of the file (right-click on the file node and select **Reload**).

It is not necessary to close the workbook or the Excel application to proceed with Design Points or Design Exploration updates. Interaction with the Excel application will be frozen during such operations but you will be able to see the performed calculations.

Limitations

The **Excel** feature in ANSYS Workbench requires Microsoft Office Excel 2007 or 2010.

The **Excel** feature is only available on Windows systems.

Troubleshooting

You are requested to install the Multilingual User Interface Pack of Microsoft Office for your language If you run the English version of Excel and the locale for the current user is configured for a language other than English, Excel will try to locate the language pack for the configured language. If the language pack is not found, then the error is reported and the automation of Excel, as used by the Excel as a Calculator feature, cannot be performed. To solve this issue, you have to install the Multilingual User Interface Pack of Microsoft Office for your language. As an alternative, you can also configure the locale for the current user to English.

This bug is documented by Microsoft. For more information, see http://support.microsoft.com/kb/320369.

Polyflow

Polyflow allows for the analysis of fluid flows with free surfaces, complex rheology (including non-Newtonian behavior with viscoelasticity), heat transfer, and chemical reactions. The usage of Polyflow involves: specifying the computation models inside Polydata, which is the module for problem setup; and running the calculations using the Polyflow solver. Polyflow also comes with several useful utilities, which can be accessed via the right-click menu on the **Setup** and **Solution** cells. For example, Polymat can be used to calculate material properties, including viscoelastic parameters based on experimental data. For more details, please see the product documentation.

There are three Polyflow fluid flow analysis systems available in Workbench:

- The **Polyflow** system provides the full simulation capabilities of Polyflow.
- The **Polyflow Blow Molding** system provides only the application-specific capabilities of Polyflow that are suited to blow molding simulations.
- The **Polyflow Extrusion** system provides only the application-specific capabilities of Polyflow that are suited to extrusion simulations.

To use a Polyflow, Blow Molding (Polyflow), or Extrusion (Polyflow) fluid flow component system, perform the following steps:

- 1. Add a Polyflow, Blow Molding (Polyflow), or Extrusion (Polyflow) fluid flow component system by dragging the system from the Toolbox into the Project Schematic, or by double-clicking the system in the Toolbox. The component system is comprised of two cells: a **Setup** cell and a **Solution** cell.
- 2. Import a mesh by right-clicking the **Setup** cell and clicking **Import Mesh** in the context menu that opens. You can merge, scale, translate, and rotate the mesh by right-clicking the **Setup** cell and clicking **Polyfuse** in the context menu that opens.
- 3. You can define your preferences for Polydata by right-clicking the **Setup** cell and clicking **Preferences** and **Polydata** in the context menu that opens.
- 4. Double-click the **Setup** cell to open Polydata, where you can specify the computational models, materials, boundary conditions, and solution parameters. Alternatively, you can import a previously saved Polyflow

data file by right-clicking the **Setup** cell and clicking **Import Polyflow Dat...** in the context menu that opens. You have the option of specifying material data by right-clicking the **Setup** cell and clicking **Polymat** in the context menu that opens.

- 5. You can define your preferences for Polyflow by right-clicking the **Solution** cell and clicking **Preferences** and **Polyflow** in the context menu that opens.
- 6. Run the Polyflow calculation by right-clicking the **Solution** cell and clicking **Update** in the context menu that opens. You can check the status of the solver during or after the calculation by right-clicking the **Solution** cell and clicking **Polydiag** in the context menu that opens. You can open the listing file to see what Polyflow has done during or after the calculation by right-clicking the **Solution** cell and clicking **Listing Viewer** in the context menu that opens.
- 7. You can generate plots of the solution data by right-clicking the **Solution** cell and clicking **Polycurve** in the context menu that opens. You can statistically postprocess the results of the solution data by right-clicking the **Solution** cell and clicking **Polystat** in the context menu that opens.

For detailed information on working with Polyflow, see the online documentation available under the **Help** menu within Polyflow. In addition, please see the separate Polyflow in Workbench User's Guide.

For information on submitting Polyflow jobs to Remote Solve Manager, see Submitting Polyflow Jobs to RSM.

Results

Use the Results component system to launch CFD-Post, a flexible, state-of-the-art post-processor that enables easy visualization and quantitative analysis of the results of CFD simulations.

Right-click the **Results** cell and select **Edit** to open CFD-Post. From CFD-Post, select **File** > **Load Results** to load a results file from the ANSYS CFX-Solver, ANSYS Fluent, or ANSYS Polyflow.

For detailed information on working with ANSYS CFD-Post, see CFD-Post in ANSYS Workbench in the CFD-Post User's Guide.

System Coupling

Use a System Coupling component system to model one- and two-way multiphysics couplings for your project. Connect a System Coupling system to one or more analysis systems, or to an External Data component, to have the System Coupling system synchronize and manage the data transfer and solution.

Drag the System Coupling component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox. Connect the System Coupling component system to the participating systems (either through the **Setup** cell or the **Solution** cell). Double-click the **Setup** cell, or right-click and choose **Edit** from the context menu to display the System Coupling tab. See System Coupling Workspace in the *System Coupling User's Guide* for further details.

Solution Cell Properties

The **Solution** cell's settings are visible from the **Properties** view. To access these settings, right-click the **Solution** cell and select **Properties**.

Most of the settings in the **Properties** view are for information only, but the **Command Line Options** setting enables you to send command-line arguments to the system coupling service. These commands are invoked when the system coupling service is started.

To learn the syntax for the command-line arguments, see "Workflows for System Coupling".

TurboGrid

ANSYS TurboGrid is a powerful tool that lets designers and analysts of rotating machinery create high-quality hexahedral meshes, while preserving the underlying geometry. These meshes are used in the ANSYS workflow to solve complex blade passage problems.

Drag the TurboGrid component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

Double-click the Turbo Mesh cell to run ANSYS TurboGrid. You can optionally connect an upstream cell to the Turbo Mesh cell to provide the geometry data. If you do not connect an upstream cell to the Turbo Mesh cell, then you can load geometry data from within the user interface of ANSYS TurboGrid.

For more information about ANSYS TurboGrid, see ANSYS TurboGrid help.

Vista AFD

Vista AFD is a program for the preliminary design of axial fans. It creates axial fan geometry data for use in BladeGen or BladeEditor. It also provides estimates of the performance of the axial fan. It may be used to generate a preliminary fan design before moving rapidly to a full 3D geometry model and CFD analysis.

Drag the Vista AFD component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

The Vista AFD component system is comprised of three cells: a Meanline cell, a Design cell, and an Analysis cell. Double-click the Meanline cell to open the cell properties, where you can specify the solution parameters, generate a solution, and view the results of a meanline analysis. If the results of the meanline calculation are satisfactory, the design (throughflow) calculation may then be performed by using the Design cell.

Optionally, before creating a BladeGen or BladeEditor model, an analysis calculation may be performed by using the Analysis cell. This uses a similar throughflow method to the design calculation but simply analyses the design created in the previous step, rather than adjusting the geometry. A significant difference between the design and analysis results indicates a potentially flawed design.

For more information about Vista AFD, see Vista AFD help.

Vista CCD and Vista CCD with CCM

Vista CCD is a program for the preliminary design of centrifugal compressors. It can be used in an iterative fashion to create a 1D design. The resulting geometry can be passed to BladeGen or BladeEditor. Vista CCD can be used to model an existing compressor and, if known, its measured performance at single operating points. An accurate 1D model can provide insight into the performance of the machine that goes beyond the test measurements.

Drag the Vista CCD or Vista CCD (with CCM) component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

The Vista CCD component system is comprised of one cell: a Blade Design cell. Double-click the Blade Design cell to open the cell properties, where you can specify the solution parameters, generate a solution, and view the results.

The Vista CCD (with CCM) component system is comprised of two cells: a Blade Design cell and a Performance Map cell. Having created a satisfactory design in Vista CCD using the Blade Design cell, you may then predict the overall performance of the designed compressor stage using the Performance Map cell.

For more information about Vista CCD, see Vista CCD help.

Vista CPD

Vista CPD is a program for the preliminary design of pumps. It creates impeller geometry data for use in BladeEditor. It may be used to generate an optimized 1D pump impeller design before moving rapidly to a full 3D geometry model and CFD analysis.

Drag the Vista CPD component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

The Vista CPD component system is comprised of one cell: a Blade Design cell. Double-click the Blade Design cell to open the cell properties, where you can specify the solution parameters, generate a solution, and view the results.

For more information about Vista CPD, see Vista CPD help.

Vista RTD

Vista RTD is a program for the preliminary design of radial inflow turbines. It can be used in an iterative fashion to create a 1D design. The resulting geometry can be passed to BladeGen, BladeEditor, and Vista TF. Vista RTD can also be used to model an existing turbine. An accurate 1D model can provide insight into the performance of the machine that goes beyond the test measurements.

Drag the Vista RTD component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

The Vista RTD component system is comprised of one cell: a Blade Design cell. Double-click the Blade Design cell to open the cell properties, where you can specify the solution parameters, generate a solution, and view the results.

For more information about Vista RTD, see Vista RTD help.

Vista TF

The Vista TF program is a streamline curvature throughflow program for the analysis of any type of turbomachine, but has been developed in the first instance primarily as a tool for radial turbomachinery analysis. The program enables you to rapidly evaluate radial blade rows (pumps, compressors and turbines) at the early stages of the design.

Drag the Vista TF component system from the Toolbox to the Project Schematic, or double-click the system in the Toolbox.

The Vista TF component system is comprised of three cells: a **Setup** cell, a **Solution** cell, and a **Results** cell. Double-click the Setup cell to open the cell properties, where you can specify the solution parameters. Double click the **Solution** cell to generate a solution. Use the **Results** cell to view the results.

For more information about Vista TF, see Vista TF help.

Custom Systems

ANSYS Workbench allows you to add custom templates and provides pre-defined custom templates, such as one-way FSI and thermal-stress coupled analyses. These pre-defined templates are provided as a convenience; you could also manually create any of these systems using system drag-drop operations, context menu operations on cells, or individual cell linking. You can also create your own templates and add them to easily build frequently-used projects.

To use one of these predefined templates, double-click the template. Drag and drop is not available as with regular templates in the Toolbox.

Available pre-defined templates include:

- FSI: Fluid Flow (ANSYS CFX) > Static Structural
- FSI: Fluid Flow (Fluent) > Static Structural
- Pre-Stress Modal
- Random Vibration
- Response Spectrum
- · Thermal-Stress

FSI: Fluid Flow (ANSYS CFX) > Static Structural

This template enables you to perform analyses that couple the physics of Fluid Flow (ANSYS CFX) and Static Structural analyses with results provided (one-way) from the former system to the latter. When this template is used, a Fluid Flow (CFX) analysis system and a Static Structural analysis system are automatically created. The two systems share a single geometry, and the fluid-flow **Solution** cell provides data that are treated as an Imported Load in the static structural **Setup** cell.

Note that similar coupled systems may also be manually created between Fluid Flow (ANSYS CFX) analysis and any of the following analysis systems:

- Transient Structural
- · Steady-State Thermal
- · Transient Thermal

FSI: Fluid Flow (Fluent) > Static Structural

This template creates two systems: a Fluid Flow (Fluent) analysis system and a Static Structural analysis system. The **Geometry** cells for the two systems share a single geometry, and the **Solution** cell in the Fluid Flow (Fluent) system provides pressure load data to the **Setup** cell in the Static Structural system.

Pre-Stress Modal

This template creates two systems: an ANSYS structural static system that transfers data into an ANSYS modal system. The two systems share **Engineering Data, Geometry**, and **Model** cells. The structural static solution provides the necessary solver files as input to the modal **Setup** cell. See the discussion on Pre Stress analysis in the Mechanical application help for more information.

Random Vibration

This template creates two systems: an ANSYS modal system that transfers data into an ANSYS random vibration system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The modal solution provides the necessary solver files as input to the random vibration setup cell. See the Random Vibration analysis discussion in the Mechanical application help for more information.

Response Spectrum

This template creates two systems: an ANSYS modal system that transfers data into an ANSYS response spectrum system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The modal solution provides the necessary solver files as input to the response spectrum **Setup** cell. See the Response Spectrum analysis discussion in the Mechanical application help for more information.

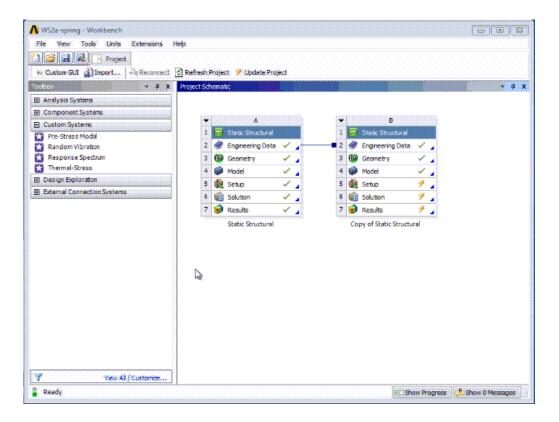
Thermal-Stress

This template creates two systems: an ANSYS steady-state thermal system that transfers data into an ANSYS static structural system. The two systems share **Engineering Data**, **Geometry**, and **Model** cells. The steady-state thermal solution provides temperature input to the static structural **Setup** cell. See the Steady-State Thermal analysis and the Initial Temperature discussions in the Mechanical application help for more information.

Adding a Custom System

To add a custom system to the Toolbox, build the system to your specifications in the Project Schematic. Then right-mouse click in the Project Schematic and choose **Add to Custom**. Type in a name for your custom system and press **Enter**. The new custom template appears in the Toolbox under Custom Systems. User-defined custom templates can also be deleted by right-clicking on the template in the toolbox and selecting **Delete**.

The following animation demonstrates adding a custom system. This animation is presented as an animated GIF in the online help. If you are reading the PDF version of the help and want to see the animated GIF, please access this section in the online help. The interface shown may differ slightly from that in your installed product.



Design Exploration

DesignXplorer provides you with the ability to perform in-depth Design Exploration studies. DesignXplorer is a powerful approach for designing and understanding the analysis response of parts and assemblies. It uses a deterministic method based on Design of Experiments (DOE) and various optimization methods, with parameters as its fundamental components. These parameters can come from any supported system, DesignModeler, and various CAD systems. Responses can be studied, quantified, and graphed. Using a Goal Driven Optimization method, the deterministic method can obtain a multiplicity of design points. You can explore the calculated Response Surface and generate design points directly from the surface or transfer data from other analysis systems or components to a Direct Optimization system.

Design Exploration systems available with ANSYS Workbench include the following. These systems will be available only if you have installed the ANSYS DesignXplorer product and have an appropriate license.

- · Direct Optimization
- Parameters Correlation
- Response Surface
- Response Surface Optimization
- Six Sigma Analysis

For an overview of these systems and links to more information about them, see What is Design Exploration?

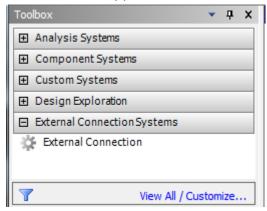
External Connection Systems

The External Connection Add-in enables you to integrate custom, lightweight, external applications and processes into the ANSYS Workbench Project Schematic workflow. Features exposed by the External Connection also allow you to perform automation and customization activities.

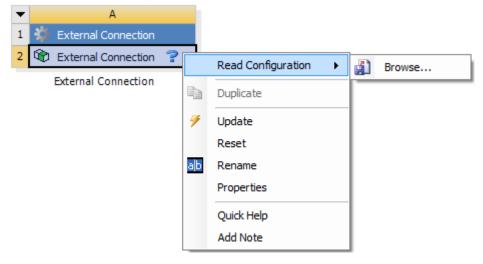
With the External Connection, you can:

- Integrate custom, lightweight, external applications.
- Define User Interface (UI) elements, such as buttons in the Workbench Toolbar or entries in custom menus, and create the scripts that enable them.
- Create new systems to facilitate interaction with the Workbench Project Schematic.

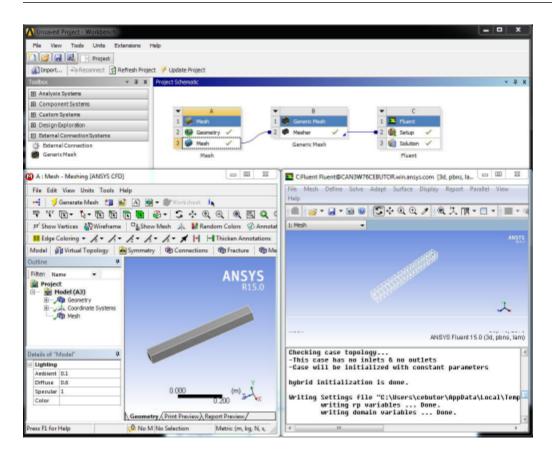
The External Connection Add-in provides an **External Connection** system in the **External Connection Systems** toolbox. The system contains a single **External Connection** component that acts as a proxy for the external application.



After you drag an External Connection cell into the Workbench Project Schematic window, the External Connection component appears in an **Edit Required** state, indicated by a question mark, until you update it with a Configuration file.



The figure below shows a Mesh Transfer system that consumes an upstream mesh and passes it to a downstream Fluent system. External Connection is used to create the *Generic Mesh* system.



For more information about the External Connection Add-in, see the External Connection Add-In.

ANSYS Workbench Interface Reference

The following user interface components are discussed in more detail:

Tabs within Workbench Views within Tabs Cells in Workbench Menus in Workbench

Tabs within Workbench

Project Tab When you open an ANSYS Workbench project, it opens to the **Project** tab. This is the main workspace where you'll interact with your project and build your analysis. By default, the **Project** tab shows the **Toolbox** view and the **Project Schematic** view.

Application and Analysis Tabs Some cells on the **Project Schematic** launch ANSYS applications that open in separate windows outside of the Workbench environment. Examples of applications that are hosted inside Workbench are Engineering Data, External Data, System Coupling, and DesignXplorer. All the cells in these systems open into separate tabs.

All the rest of the cells on the **Project Schematic** open tabs inside Workbench: tabs for ANSYS applications that are hosted in Workbench, and tabs corresponding to cells representing specific analysis steps. Examples of analysis steps (cells) that open tabs are the **Parameter Set** bar, the **Parameter Set** cell, and the **Analysis** cell in a Mechanical APDL or Excel component system.

Views within Tabs

ANSYS Workbench provides the following categories of views:

Project Schematic View Common Views Persistent Views

Project Schematic View

Located on the **Project** tab, the **Project Schematic** view is the main workspace for your project

Project Schematic Properties

The **Properties** view of the **Project Schematic** has the following properties:

Notes

This property displays project notes created via the Project Schematic **Add Note** context menu option. For more information on adding project notes, see Project Schematic Context Menu Options (p. 271).

Update Options

This property allows you to specify whether the **Update Project** action will be performed as a job submitted to the Remote Solve Manager

When this property is set to **Submit to Remote Solve Manager**, the related **Solve Manager**, **Queue**, **Pre-RSM Foreground Update**, and **Component Execution Mode** properties become enabled.

When the project is sent to RSM via the **Update Project** option, only those systems above the **Parameter Set** bar will be submitted updated remotely.

For more information, see Submitting Projects to Remote Solve Manager (RSM) for Remote Update (p. 49).

Common Views

Common views are ones that are included on multiple tabs, but are configured per tab (i.e., changes to the view on one tab are not reflected on other tabs of the same sort, because the view contains different data on each tab).

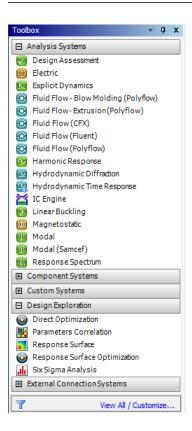
The following common views are available:

Toolbox View
Toolbox Customization View
Files View
Outline View
Properties View
Table View
Chart View
Scene View
Solution Information View

Toolbox View

On the **Project** tab, the **Toolbox** contains the different types of systems you can add to the **Project Schematic**. Systems are divided into categories which can be expanded or collapsed to show or hide the systems available in that category. You can select systems from the following system categories:

Analysis Systems (p. 135) Component Systems (p. 149) Custom Systems (p. 240) Design Exploration (p. 242) External Connection Systems (p. 243)



For more information,, see Systems.

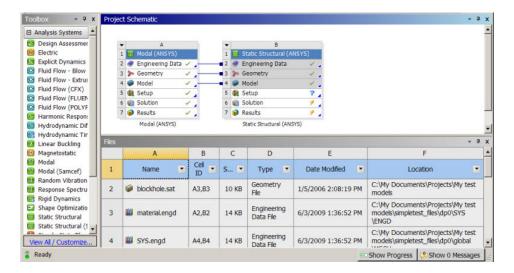
Toolbox Customization View

The **Toolbox Customization** view allows you to specify which systems display in the **Toolbox**. For more detailed information on this view, see Customizing the Toolbox on the Project Tab (p. 9).

Files View

The **Files** view shows a list of all files associated with the project. It allows you to see the name and type of file, the ID of the cell(s) the file is associated with, the size of the file, the location of the file, and other information. You can sort the list via drop-down menus in the column headers.

Files added to the project will appear here. Files missing or deleted from the project will be shown in red and will be marked with a "Deleted" icon. To remove deleted files from the **Files** view, right-click on the line containing the deleted file and select **Remove** filename from **List** from the context menu. Use the **Ctrl** key to select multiple lines. See Project File Management for more information on working with missing or deleted files.



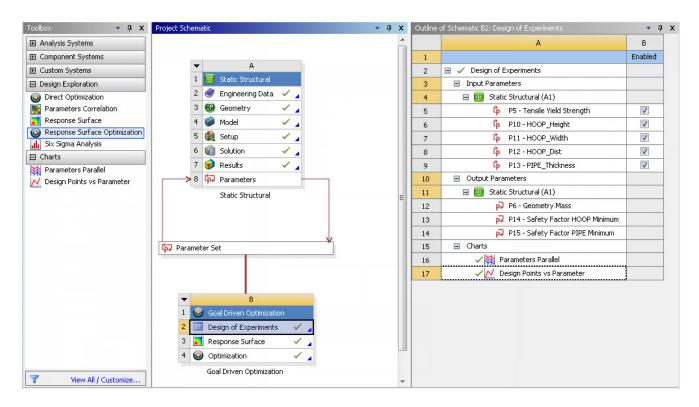
Right-click any of the cells and select **Open Containing Folder** to open your operating system's file manager to the folder containing that file. In most cases, you should not edit, add, or delete a file from the operating system file manager. ANSYS Workbench will not recognize or be aware of any changes that you make directly in the file system (such as adding or removing a file). However, if used with caution, this view can be a useful way to edit files such as application input files (for example, the Mechanical APDL application input files).

Right-click any of the cells and select **File Type Filter** to choose which types of files you want to appear in the **Files** view.

Right-click one or more of the cells and select **Copy** to copy the text in the selected cell(s).

Outline View

When data is available for an item, the **Outline** view presents data in an outline form. You can access the **Outline** view on the **Project** tab, the **Parameter Set** tab, the **Parameters** tab, or the tab for any cell in a Design Exploration or Engineering Data system. To do so, double-click the **Parameter Set** bar or a system cell, or by right-clicking and selecting **Edit** from the context menu.



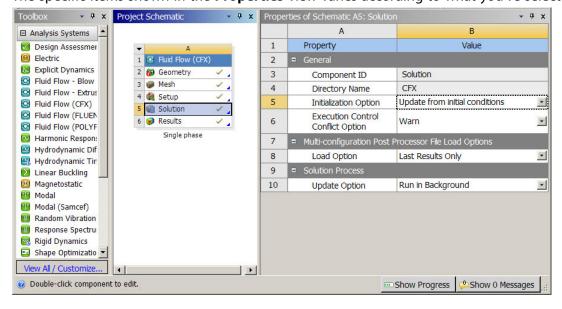
Examples of the **Outline** view include the **Outline of All Parameters** on the **Parameter Set** tab and the **Outline of Schematic** on an application tab.

For detailed information about using the **Outline** view for parameters and design points, see Working with Parameters and Design Points (p. 93).

Properties View

The **Properties** view allows you to see properties for the **Project Schematic**, systems, cells, and schematic links.

The specific items shown in the **Properties** view varies according to what you've selected.



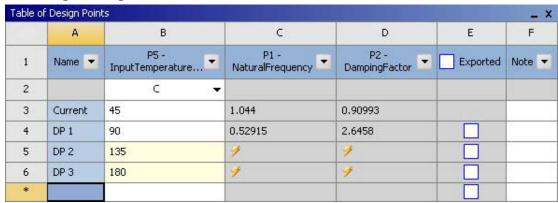
In some cases, it displays basic information about the component; this information cannot be modified from the **Properties** view. For example, if you choose to view properties on a link between systems, you will see a short list of uneditable connection properties, including type and the from/to cell identifiers.

In other cases, the **Properties** view displays detailed information, some or all of which is editable. For example, if you choose either **Edit** or **Properties** from the Vista TF **Setup** cell, you see a fully-editable list of properties.

If more detailed information a specific **Properties** view is available, view the Quick Help for the cell for which you are viewing properties.

Table View

When the **Table** view is included on a tab, it allows you to view project data in table format. Examples of the **Table** view are the **Table of Design Points** on the **Parameter Set** tab and the **Table of Properties** on an **Engineering Data** tab.



Some tables are editable and some are not. In general, if you can add a new row, you will see an asterisk (*) in the last row. If a cell is editable, you will be able to double-click in the cell to select the content and change it. In some tables, columns may include a drop-down menu of additional actions that are available for that column. Tables for different applications and different purposes will have different features and behavior. See your application documentation for details on using tables in your specific application.

Chart View

When the **Chart** view is included on a tab, it allows you to view charts that have been added and generated for the project (you can add charts or new chart instances from the **Toolbox**). Each type of tab has different charts available; for example, the **Parameter Set** tab will have a different set of charts than a DesignXplorer **Parameters Correlation** tab does.

For more detailed information about using charts, see Working with the Chart View (p. 81).

Scene View

The **Scene** view is available in the System Coupling or Fluent system's interface. For more information, see System Coupling User's Guide or Fluent in Workbench User's Guide.

Solution Information View

The **Solution Information** view is available in the System Coupling or ANSYS Fluent system's interface. For more information, see System Coupling User's Guide or Fluent in Workbench User's Guide.

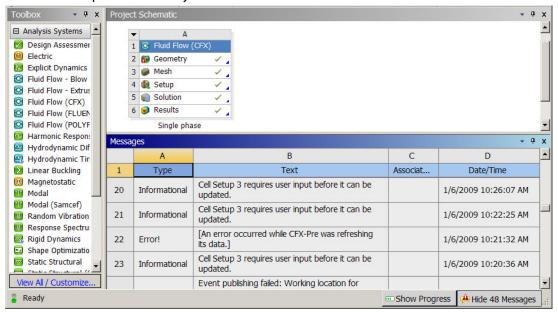
Than

Persistent Views

Some views are persistent across the all the tabs. The **Messages**, **Progress**, and **Sidebar Help** views remain as you've specified as you navigate between tabs. For example, if you choose to show the **Progress** view while in the **Project** tab, it will remain visible as you move through other open tabs.

Messages View

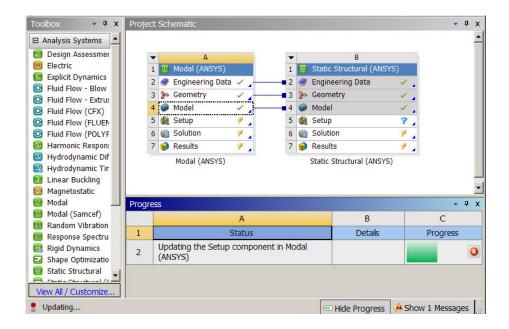
The **Messages** view displays any messages you have, such as error and warning messages, solver messages, status messages, and so on. During any update (cell, system, project, design point, multiple design point), any messages that are generated will be displayed in the **Messages** view. The **Messages** view will open automatically if an error message occurs during an update, but in other situations you will need to open it manually.



You can display or hide the **Messages** view by clicking the **Show Messages** or **Hide Messages** button in the lower right corner of any ANSYS Workbench tab.

Progress View

The **Progress** view displays a progress bar during an update. You can display or hide the **Progress** view by clicking the **Show Progress** or **Hide Progress** button in the lower right corner of any ANSYS Workbench tab. To interrupt an update process, click the Interrupt button [] of the **Progress** view. Be aware that not all processes can be interrupted, and some processes that are interruptible may have periods where they cannot be interrupted. Because some processes can stop cleanly only at certain checkpoints, the delay between the interrupt request and the actual interruption can sometimes be lengthy.



Note

For Mechanical runs, only Static Structural (Static Structural analysis using ANSYS solver) and Transient Structural (Transient Structural analysis using ANSYS solver) support interrupt.

Cells in Workbench

The systems that you add to the **Project Schematic** are made up of one or more cells.

The following cells topics are discussed below:

Types of Cells Understanding Cell States Cell Properties

Types of Cells

The following common types of cells occur in many of the analysis and component systems available in ANSYS Workbench; how you work with them is explained below. Other cell types may be available in certain systems; see the application-specific documentation under Systems for these cell descriptions.

Engineering Data Geometry Model/Mesh Setup Solution Results

Engineering Data

Use the **Engineering Data** cell with Mechanical systems or the Engineering Data component system to define or access material models for use in an analysis. To define material data, open the **Engineering Data** tab by either double-clicking the **Engineering Data** cell or right-clicking the cell and selecting **Edit** from the context menu. For more information, see Engineering Data.

Geometry

Use the **Geometry** cell to import, create, edit or update the geometry model used for analysis. Right-click on the cell to access these functions via the context menu. The right-click options are context sensitive and change as the state of your geometry changes, so not all of the geometry-specific options are described here will be available at all times. These options are in addition to the common options described in Common Context Menu Options (p. 265) and Transfer Context Menu Options (p. 268).

Note

The SpaceClaim Direct Modeler features are available only if you have SpaceClaim Direct Modeler installed and the ANSYS SpaceClaim Direct Modeler license available.

New Geometry (or New DesignModeler Geometry/New SpaceClaim Direct Modeler Geometry) Launches DesignModeler or SpaceClaim Direct Modeler, where you can build a new geometry.

Import Geometry

Select **Browse** to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files.

Edit (or Edit Geometry in DesignModeler/Edit Geometry in SpaceClaim Direct Modeler)

After you have attached a geometry to your system by choosing either **New Geometry** or **Import Geometry**, click **Edit** to open the model in DesignModeler or SpaceClaim Direct Modeler to modify it.

Replace Geometry

Select **Browse** to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files to replace the currently specified file.

Update from CAD

Generates an existing CAD geometry using the parameter values as defined in the CAD system.

Refresh

Reads in all modified upstream data but does not regenerate the geometry. Enabled when the Geometry cell is in the Refresh Required state.

Properties

Displays a **Properties** view where you can select basic and advanced geometry properties. For a detailed description of the options available from the **Properties** view, see Geometry Preferences in the CAD Integration section of the ANSYS Workbench help.

Model/Mesh

The **Model** cell in the Mechanical application analysis systems or the Mechanical Model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition.

When linking two systems, you cannot create a share between the Model cells of two established systems. You can generate a second system that is linked at the **Model** cell of the first system, but you cannot add a share after the second system has been created. Likewise, you cannot delete a link between the **Model** cells of two systems.

The **Mesh** cell in Fluid Flow analysis systems or the Mesh component system is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

Edit

Launches the appropriate Model or Mesh application (the Mechanical application, Meshing, and so on.)

Setup

Use the **Setup** cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS Workbench, including connections between systems.

For the Mechanical application systems, you will see the following **Setup** options, in addition to the common options:

Edit

Launches the Mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the Mechanical application.

For CFX systems, you will see the following Setup options, in addition to the common options:

Edit

Launches CFX-Pre.

Import Case

Imports an existing case file containing physics data, region and mesh information for your analysis.

For Fluent systems, you will see the following Setup options, in addition to the common options:

Edit

Launches ANSYS Fluent.

Import Case

Imports an existing Fluent case file.

Solution

From the **Solution** cell, you can access the Solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, you will see the Solution cell in a pending state until the remote process completes. See the discussion on Understanding Cell States (p. 255), below.

For the Mechanical application systems, you will see the following Setup options, in addition to the common options described earlier:

Edit

Launches the Mechanical application open to the Solution branch.

Delete

Deletes the **Solution** and **Results** cell. Deleting the solution cell makes the system a setup-only system, meaning the system will generate only an input file. It will not solve or post results. The Solution object and below are removed from the Mechanical application tree.

For CFX systems, you will see the following Solution options, in addition to the common options:

Edit

Launches CFX-Solver Manager.

Import Solution

Displays the most recent CFX-Solver Results files imported (if any) and enables you to browse for such files using the **Open** dialog box, where you can specify the CFX-Solver Results file to load. When the results file is loaded, the system will display only the **Solution** cell and the **Results** cell.

Display Monitors

Opens the ANSYS CFX-Solver Manager and shows the results of the previous run.

For Fluent systems, you will see the following Solution options, in addition to the common options:

Edit

Launches ANSYS Fluent.

Import Final Data

Allows you to select an existing Fluent data set (e.g., solved on an external cluster) into a **Solution** cell in a Fluent system and immediately start post-processing in CFD-Post, without the need to run the minimum of one more solver iteration. This option becomes available after importing case file into the **Setup** cell.

Results

The **Results** cell indicates the availability and status of the analysis results (commonly referred to as postprocessing). From the **Results** cell, you cannot share data with any other system.

Understanding Cell States

ANSYS Workbench integrates multiple applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. As a result of this flow of data, a cell's state can change in response to changes made to the project. ANSYS Workbench provides visual indications of a cell's state at any given time via icons on the right side of each cell.

Cell states can be divided into the following categories:

Typical Cell States Solution-Specific States Failure States

Typical Cell States

Unfulfilled



Required upstream data does not exist. Some applications may not allow you to open them with the

cell in this state. For example, if you have not yet assigned a geometry to a system, all downstream cells will appear as unfulfilled, because they cannot progress until you assign a geometry.

Refresh Required



Upstream data has changed since the last refresh or update. You may or may not need to regenerate output data. When a cell is in a refresh required state, you generally have several options:

• You can edit the cell and choose to review the unrefreshed data.

- You can refresh the data, which will read the upstream data, but will not perform any long running operation. For instance, if the geometry changed, thus placing the Mesh cell in a Refresh Required state, a refresh on the mesh cell would update the geometry without generating a new mesh (which could potentially be a lengthy operation).
- Update the cell, which would refresh the data and regenerate any output data.

The advantage to simply refreshing a cell rather than performing a full update is that you can be alerted to potential effects on downstream cells before updating and can make any necessary adjustments. This option is especially useful if you have a complex system in which an update could take significant time and/or computer resources.

Attention Required



All of the cell's inputs are current; however, you must take a corrective action to proceed. To complete the corrective action, you may need to interact with this cell or with an upstream cell that provides data to this cell. Cells in this state cannot be updated until the corrective action is taken.

This state can also signify that no upstream data is available, but you can still interact with the cell. For instance, some applications support an "empty" mode of operation, in which it is possible to enter the application and perform operations regardless of the consumption of upstream data.

Update Required



Signifies that local data has changed and the output of the cell needs to be regenerated. When updating a Refresh Required cell, the Refresh operation will be performed and then the Update operation will be performed.

Up to Date



An Update has been performed on the cell and no failures have occurred. It is possible to edit the cell and for the cell to provide up-to-date generated data to other cells.

Input Changes Pending



Indicates that the cell is locally up-to-date but may change when next updated as a result of changes made to upstream cells.

Solution-Specific States

In addition, the **Solution** or **Analysis** cell for certain solvers may support the following solution-specific states.

Interrupted

Indicates that you have interrupted the solution. This option performs a graceful stop of the solver, which will complete its current iteration and write a solution file. You can use that solution for postprocessing (to look at the intermediate result, for example). You could also elect to continue to solve from that point using the Resume or Update function.

Pending



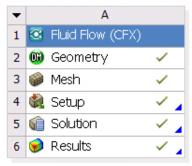
Signifies that a batch or asynchronous solution is in progress. When a cell enters the Pending state, you can interact with the project to exit ANSYS Workbench or work with other parts of the project. If you

make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

Note

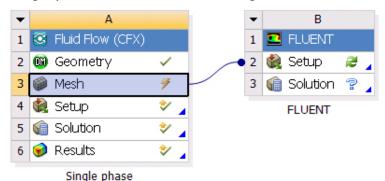
When up-to-date cells are connected to cells in a *different type of system*, the state of the up-to-date cells may change to update required. This behavior occurs because additional files have to be generated to satisfy the newly added system.

For example, if you build a Fluid Flow (ANSYS CFX) system that is up-to-date, as shown below,



Single phase

and then connect the mesh to a Fluid Flow (Fluent), the mesh files for CFX Setup are the only files that will have been generated by the update process. After connecting another system to the up-to-date **Mesh** cell, the **Mesh** cell transitions to the Update Required state to signify that additional files must be generated in order to satisfy the Fluent system.



The cells already connected to the **Mesh** cell will transition to the Refresh Required state after the **Mesh** cell is updated, and updating the project will unnecessarily re-update the previously up-to-date cells (CFX Setup, Solution, and Results). As such, if this is the desired project structure, we recommend that you establish the connection to the Fluent system before updating components downstream of the Mesh in the Fluid Flow (ANSYS CFX) system in this example.

Failure States

If a particular action fails, ANSYS Workbench provides a visual indication as well. Failure states are described below.

Refresh Failed, Refresh Required

The last attempt to refresh cell input data failed and the cell remains in a refresh required state.

Update Failed, Update Required

The last attempt to update the cell and calculate output data failed and the cell remains in an update required state.

Update Failed, Attention Required

The last attempt to update the cell and calculate output data failed. The cell remains in an attention required state.

If an action results in a failure state, you can view any related error messages in the **Messages** view by clicking the **Show Messages** button on the lower right portion of the ANSYS Workbench tab.

Cell Properties

Every cell in ANSYS Workbench has various properties associated with it. Most of these properties are specific to the type of system, type of cell, and state of the cell, and are generally documented with the specific application documentation. However, some cell properties are common across many cells, regardless of type of system or cell state. Those properties are described here.

To view cell properties, right-click on the cell and select **Properties** from the context menu. The **Properties** view will open, showing all properties applicable to that cell in its current state. Both the property name and its current value are shown. Some properties are editable, while others cannot be edited because of the specific configuration of that project and/or cell.

Common Cell Properties

The cell properties described here are common to most cells. For information on additional cell properties, see the appropriate application documentation.

Component ID

This property shows the name of the component.

Directory Name

This property shows the directory where any information associated with this component resides. For more information on the general ANSYS Workbench directory structure, see Project File Management (p. 71).

Notes

This property displays system and cell notes created via the **Add Note** context menu option. For more information on adding notes, see Common Context Menu Options (p. 265).

Last Update Used Licenses

This property shows the license used by this component during the most recent update. If the component does not require a license during an update, the value will be Not Applicable.

Always Include in Design Point Update

This property is used for External Connection and CFD-Post components. It allows you to specify that a component should be included in any design point update operation, even if it does not have output parameters. If you enable this property for a component, the component will be updated when you update design points, regardless of whether parameter values will be affected.

Menus in Workbench

The following types of menus are available in Workbench:

Menu Bar Context Menus

Menu Bar

The **Menu Bar** gives you access to the following menus:

File Menu

View Menu

Tools Menu

Units Menu

Extensions Menu

Help Menu

File Menu

The **File** menu, you can manage your project files:

New

Open a new project. If you already have a project open, this action will close the current project.

Open

Open an existing project that had been saved previously. If you already have a project open, this action will close the current project.

Save

Saves the current project. If the current project has already been saved, this action will save any changes to the current location. If the current project has not already been saved, you will be prompted to specify a name and location for the file.

Note

You can also save the current project via the **Save** icon underneath the **File** menu.

Save As

Saves the current project under a different name and/or location. You will be prompted to specify the name and location for the file.

Save to Repository

Saves the current project to an EM repository. You must have saved the project in ANSYS Workbench before you can save it to the EKM repository. In addition to the name, you can include a brief description of the project. This description will be visible when you view it in the EKM repository. You can also choose whether to include results and/or external files when you save the project by using the selections under the **Show Options** drop-down. See Working with ANSYS Workbench and EKM (p. 119) for more information.

If you have not previously established a connection to an EKM repository, you will be presented with a connection wizard that will allow you to set up a connection. You must have a connection already established in order to save a project to a repository.

Open from Repository

Opens a project saved in an EKM repository. You can also save a working copy to a local directory. See Working with ANSYS Workbench and EKM (p. 119) for more information.

If you have not previously established a connection to an EKM repository, you will be presented with a connection wizard that will allow you to set up a connection. You must have a connection already established in order to save a project to a repository.

Send Changes to Repository

Sends changes made to a previously-saved project to the EKM repository. This option is available only if the local copy is newer than the repository copy. You can explicitly select this option, or ANSYS Workbench will remind you to send local changes to the repository when the project is about to replaced.

Get Changes from Repository

Retrieves changes made to a project that has been previously opened from or saved to the EKM repository. If a more recent version than your local copy exists in the repository, you will be prompted and given the option to update your local copy with the version of the project in the repository. You can explicitly select this option, or ANSYS Workbench will prompt if you wish to get changes from the repository.

Manage Repository Project

Enables access to features that monitor or change the status of a project in the repository. The permissions you have for a project in a repository control which features you can access.

Tip

To view your permissions for a project in the repository, right-click the project and select **Display** > **Permissions**.

Register Session with Repository

Allows you to register your current Workbench session with the EKM repository. Once the session has been successfully registered, a job is created for it in the EKM **My Jobs** folder. From the **My Jobs** folder, you can perform and monitor various Workbench operations remotely.

Check Out/Undo Checkout

Controls whether you have the project checked out from a version control system.

Add to Version Control/Remove from Version Control

Controls whether the project is under version control. When a project is under version control, changes can be made by only one user at a time.

Get Exclusive Control/Release Exclusive Control

Controls whether you have the exclusive use of a repository project that is not under version control.

Access Control Status

Displays the **Access Control Status** dialog box, which displays the version control status, the name of the person who currently has the file checked out (if any), the version number, the name of the user

whose check-in last modified the file's version number, the date the file was last modified, and any stored comments.

Important

When you display the **Access Control Status** dialog box, the **Manage Repository Project** menu items are refreshed with the most recent information from the repository.

Launch EKM Web Client

Select this option to launch the EKM login screen to access the EKM workspace. EKM will display in your default browser using a currently established connection. If you already have connections established to more than one repository, you will be asked to select the connection to use. If you do not already have a connection established, you will see an error message. Choose **Open from Repository** to launch the EKM connection wizard to establish a connection for the first time.

Save to Teamcenter

Teamcenter is a software package designed for computer-aided product data management. Use this option to save the current project into the Teamcenter database. You can then open the project dataset in ANSYS Workbench. For more information on using the ANSYS Teamcenter connection, see the Cad Integration section of the ANSYS Workbench help.

Import

Imports a legacy database and converts it to the appropriate systems in the current project. You can also use this option to assemble multiple legacy databases into a single project.

Archive

Generates a single archive file that contains all project files. This archive will include the project file and all files in the project name_files directory with a few optional additions/exceptions that you can specify:

- result/solution items
- imported files external to the project
- items in the user files folder

The archive will be saved as a Workbench Project Archive (.wbpz) or a Zip (.zip/.tar.gz) file.

Previously imported external files from a restored archive directory are treated as internal files if archived again.

See Archiving Projects (p. 75) for details on archiving projects.

Restore Archive

Restores a previously-generated archive file. After you select the project archive to be restored, you will be prompted for the name and location where the restored file(s) are to be located. After the archive is extracted, the project will open in ANSYS Workbench.

You can also extract the archive manually by using an unzip utility, and then opening the .wbpj file.

Scripting

Use this option to record a journal of your session, execute a journal or script, or open a Python command window. Choose one of the following options:

- Record Journal: Creates a journal of the ANSYS Workbench session.
- **Stop Recording Journal**: Stops recording of the current session.
- Run Script File: Runs a previously-created journal or script.
- Open Command Window: Opens a Python command window for issuing ANSYS Workbench commands.

You can also select from a list of previously-used journals.

For more information on recording journals and creating scripts for reuse, see Using Journals and Scripts (p. 69).

Export Report

Select this option to write out a report of the current project in .html/.htm format. The report will be written to the user_files directory under the project directory by default. You can control whether the report opens by default using the **Options>Project Reporting** settings.

The report contains basic project information, including:

- · Export time and date
- ANSYS Workbench version number
- a graphic of the systems as shown in the project schematic
- file information
- · parameter and design point information
- system and cell information

The specific information provided will vary depending on the contents of the project. Additional information may be available from the individual applications. Not all applications provide reporting information.

Recently Viewed Files

Shows the four most recently-opened projects.

Exit

Exits and closes ANSYS Workbench. You will be prompted to save any unsaved data.

View Menu

The **View** menu provides the following options for controlling the window layout:

Refresh

Updates the view.

Reset Workspace

Restores the current workspace layout to default settings.

Reset Window Layout

Restores the original window layout.

List of Views

All ANSYS Workbench views are listed. You can select which view(s) you want to display. Any changes you make to views are carried forward to subsequent ANSYS Workbench sessions. For a detailed description of the individual views listed here, see Views within Tabs (p. 245).

Show Connections Bundled

Shows multiple links connecting systems as a single link where possible. A bundled connection includes a label indicating the shared cells (e.g., "2-4" indicates that cells 2, 3, and 4 are shared between systems)

Show System Coordinates

Shows the alphanumeric column and row headings for each system. This option is selected by default.

Tools Menu

The **Tools** menu provides the following project and user preference option:

Reconnect

Reconnects to updates that were pending when the project was closed. This option is available only if the project has cells in the Pending state.

After reconnecting to pending solution data, it is important to save the project. For a **Solution** cell update: If you decline to save the project before exiting, the intermediate solver data will be discarded and will not be accessible in future ANSYS Workbench sessions. For more detailed information, see Exiting a Project during an RSM Solution Cell Update (p. 68).

Refresh Project

Refreshes all cells in the project that are in a Refresh Required state.

Resume

Resumes design point updates that were pending when the project was closed. This option is available only if the project has design points that are paused due to cells in a Pending state.

Update All Design Points

Updates all design points in a project that are in an Update Required state. This option is available only when the project contains multiple design points.

Update Project

Updates all cells in the project that are in an Update Required state.

Note

If you use this option and the project is being updated remotely via RSM, only the systems and cells above the **Parameter Set** bar will be submitted to RSM. If needed, DesignXplorer systems can be further updated once the remote project update is completed. For more information, see Submitting Projects to Remote Solve Manager (RSM) for Remote Update (p. 49).

Abandon Pending Updates

Use this option if you have a remote or background solve process (some component or components in the project were saved in a Pending state) and attempts to reconnect have failed. This option will ignore any results calculated thus far and return the project its normal state. If you use this option, you may need to manually remove processes that were not stopped or files that were not removed.

License Preferences

Opens the **License Preferences for User** dialog box. Use this dialog box to specify which licenses at your site you want to be able to use, and to specify which licensing method to use. For details on using the **License Preferences for User** dialog box, click **Help** in the dialog box or see Using Software Licensing in ANSYS Workbench (p. 28).

Release Reserved Licenses

Use this option to manually release a reserved license if a job hangs or a reserved license is not released normally (for example, if you delete a project that contains pending updates using reserved licenses). Select the project for which you want to release licenses, and click **Release Selected**. This feature is intended only as a license recovery method and should not be used in normal operations. See Reserving Licenses for a Design Point Update (p. 114) for more information about using reserved licenses for a design point study.

Launch Remote Solve Manager

Launches the Remote Solve Manager (RSM) interface. With the RSM user interface, you can filter jobs by status, manage queues and servers of local and remote solve managers, monitor the progress of jobs, and delete jobs. For detailed information on running RSM, see the Remote Solve Manager User's Guide.

Options

Defines your preferences for ANSYS Workbench. The preferences you set here are local settings and affect only you. For detailed descriptions of the **Options** settings, see Setting ANSYS Workbench Options.

Units Menu

The **Units** provides the following options for specifying unit systems:

Display Values as Defined

The value and unit as defined in ANSYS Workbench or the original source application will be displayed. No conversion information is displayed.

Display Values in Project Units

The value will be converted for display to correspond to the selected project unit system.

Unit Systems

Allows you to access the **Unit Systems** dialog, where you can choose to display any or all of the predefined unit systems. Quantity names and units for each of the predefined unit systems is also shown. You can also define custom unit systems as well. For custom unit systems, you can define the units for each quantity name. You can display up to 15 unit systems (any combination of predefined or custom) at a time.

For detailed information on working with units in ANSYS Workbench, see Configuring Units in Workbench (p. 10).

Extensions Menu

The **Extensions** menu provides options for managing the extensions developed for ANSYS products using the ANSYS Workbench Customization Toolkit (ACT).

Manage Extensions

Displays the **Extensions Manager** tool, which allows you to specify which extensions will be loaded.

Build Binary

Compiles a binary version of an extension from the scripted version.

Install Extension

Installs the extension into your Application Data folder and ensures that the extension is available in the **Extensions Manager**.

View Log File

Opens the extension log file, which provides warning or error messages generated by the extension.

For more detailed information on these menu options, see the ANSYS Workbench Customization Toolkit Developer's Guide in the ANSYS Customization Suite.

Help Menu

The **Help** menu provides options that allow you to access help for ANSYS Workbench or to access help for most ANSYS, Inc. products, including Installation and Licensing help. You can also view context-sensitive help and view the version information.

Context Menus

Context menu options provide capabilities that enable you to work with your existing systems or to add to and modify projects.

The following types of context menus are available in Workbench:

Common Context Menu Options
Transfer Context Menu Options
Tab Context Menu Options
System Header Context Menu Options
Project Schematic Context Menu Options
Link Context Menu Options

Common Context Menu Options

In addition to the menu items that are unique to each system or cell, some of the options on the contextsensitive menu are available with most systems and cells. They include:

Duplicate

Creates a new system that is a duplicate of the selected system. All data associated with unshared cells in the system is copied to the duplicate system: all data above the cell from which the duplicate operation was initiated is shared; data at and below the cell from which the duplicate operation was initiated is copied. See Moving, Deleting, and Replacing Systems (p. 48) for more information on duplicating systems.

Update

Refreshes input data (see Refresh, below) and generates required output data. Any upstream cells upon which the cell is dependent will also be updated. Update is not possible on a cell if another update is currently running on the cell or on any of the upstream cells. In some cases, the Update option may still be available while other operations are in progress, such as Clear Generated Data, Reset, Delete, or Duplicate. If those operations are in progress on a separate system, you can safely update the current system. However, if those operations are in progress on the same system that you wish to update or on a system connected to the one you want to update, you must wait until those operations are completed before beginning an update.

If a project contains multiple design points, this option updates only the current design point.

If a system contains a coupled cell, the Update option on the coupled cell will be disabled. To update the system that contains a coupled cell, you must update the coupled system so that both the coupled cell and all cells dependent on the coupled cell can be updated.

If a project is unsaved and any cells in your project are configured to use RSM or run in the background, you must save the project or change the solution process settings of those cells to run in the foreground before you can update.

Rename

Renames the system or cell.

Refresh,

Reads in all modified upstream data but does not necessarily regenerate the outputs of the cell.

In some cases, the Refresh option may still be available while other operations are in progress, such as Clear Generated Data, Reset, Delete, or Duplicate. If those operations are in progress on a separate system, you can safely refresh the current system. However, if those operations are in progress on the same system that you wish to refresh or on a system connected to the one you want to refresh, you must wait until those operations are completed before beginning a refresh.

Quick Help

Displays a quick help panel for the cell, if available. Quick help provides a brief description of how to use the cell in its current state. You can also left or right-click on the blue triangle (where available) in the lower right corner of a cell to view quick help.

Add/Edit Note

Displays an editable panel where you can enter notes about a system or cell. There is no limit to the amount of text you can type into a note; as you type, the panel increases in length. You can also edit an existing note by left- or right-clicking on the green triangle in the upper right corner of a system or cell, or by editing the **Notes** field in the **Properties** view. To close the note, click outside the panel. To delete a note, delete the text within the panel or from the **Properties** view.

The content of notes is also included in project reports.

Properties

Displays applicable cell properties in the **Properties** view.

Recently Used

Lists all recently used files.

Two additional items that are available with most context menus are:

Clear Generated Data

Removes or erases any data that the cell has generated or is to generate and store if any such data is present. This data can include mesh, input files, solver files, and so on. Clear may alter the state of the current cell and cells downstream from the selected cell.

Reset

Removes or erases both input and output data to the cell and sets the cell state back to default. Any reference files are removed. Reset may alter the state of cells downstream from the selected cell. If you

have two systems that share cells, reset is not available from the cells that are shared, only from the source (the cell that is being shared from).

Important

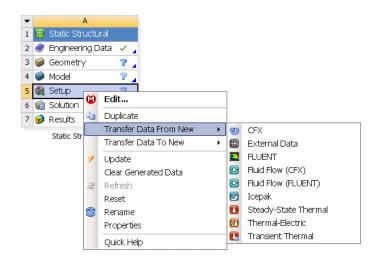
For some systems, these two context menu options result in specific behavior depending on the selected cell and in some cases specific system data (such as existing links). Refer to the following table for specific actions.

Cell	Data	Clear Generated Data	Reset
Mechani	ical systems		
Geo- metry			Clears the geometry source and rests geometry properties to defaults.
Model		Deletes mesh, input files, and solver files. Cleans results.	Closes the Mechanical session (if open) and deletes the .mechdb from disk. System is in a state as if the geometry was never attached.
Setup		Deletes input and solver files. Cleans results.	Deletes any objects under the Environment (like loads and supports).
Solu- tion		Deletes solver files. Cleans results. Input files (such as ds.dat, CARep.xml) are left intact.	Set values in "Analysis Settings" back to defaults.
Results		Cleans any solved results. Solver files are left intact.	Deletes any results, probes, or post tools from Mechanical.
Setup	Imported Temper- ature: One- way FSI input	Cleans imported load and deletes transfer xml file. User specifications (such as scoping) are intact.	Deletes the Import Load from Mechanical and also deletes CFD result file from disk. State on Setup cell goes to Refresh Required.
Setup	Imported Temper- ature: Thermal stress	Cleans imported load and deletes transfer xml file. User specifications (such as scoping) are intact.	No additional action required.
Model	Link to FE Model- er	Deletes SYS.cmdb.	No additional action required.
Model	Link to CFX	Deletes SYS.cmdb.	No additional action required.
Model	Link to Mechanic- al APDL	Deletes SYS.cdb.	No additional action required.
Setup	Link to FE Model- er	Deletes .mechdb.	No additional action required.
Setup	Link to Mechanic- al APDL	No additional action required (ds.dat is the natural output from the Setup cell).	No additional action required.
Solu- tion	Link to Mechanic- al APDL	No additional action required (file.rst is the natural output from the Solution cell).	No additional action required.
Mechani	ical APDL	•	

Cell	Data	Clear Generated Data	Reset
Analys- is		Deletes ANSYS-generated files (created during Update or Edit). Files added via input links or Add xxx File are left intact.	Deletes all files in the Mechanical AP-DLSystem directory and resets any properties (as set in the Property view) back to default. Any schematic input links remain intact and the needed files are copied back into the system upon Refresh. Any manually- added files (Add Input/Reference) will be deleted and removed from Outline view.
FE Mode	ler		
Model		No additional action required.	Deletes . fedb and sets any properties (Properties view) back to default.
Model	Link to Engineer- ing Data	Deletes FiniteElementMod-elMaterials.xml.	No additional action required.
Model	Link to Mechanic- al Model cell	Deletes FEModeler- File.rsx, FEModeler- File.fedb, ACMOFile.dat.	No additional action required.
Model	Link to Geometry	Deletes Parasolid- File.x_t.	
Model	Link to Mechanic- al APDL	Deletes ANSYSInput-File.inp.	
Fluent			
Setup	Link to imported case and other input files, set- tings, and pos- sibly other input files		Closes Fluent session without saving data. Deletes all internal files. Links to imported files are deleted (but file is not deleted.) If Mesh is coming from an upstream simulation Mesh cell, the file is unregistered and might get deleted if the upstream Mesh cell no longer refers to it. All associated input parameters are deleted. Launcher settings are set to default value.
Solu- tion	Internal, link to initialization data	Closes session without saving data. Deletes all files currently associated with the cell (latest available solution data). Any schematic input links will remain intact.	Closes session without saving data. Deletes all files currently associated with the cell (latest available solution data). Any schematic input links will remain intact. Any imported initial solution data file will be unregistered. (Only link is removed; file is not deleted.) All associated output parameters are deleted. Launcher settings are set to default value.

Transfer Context Menu Options

In addition, many cells will also include transfer options in their context menu:



Transfer Data from New...

Creates an upstream system that can provide data to the selected cell. Only those systems that can provide valid data to the selected cell are shown. When you choose a system from the options shown here, that system will appear to the left of the currently-selected system, with all appropriate connections drawn.

Transfer Data to New...

Creates a downstream system that can accept data from the selected cell. Only those systems that can accept data from the selected cell are shown. When you choose a system from the options shown here, that system will appear to the right of the currently-selected system, with all appropriate connections drawn.

Transfer Data options are available only from root cells. If a cell is derived from an upstream cell (for example, when two systems share a geometry), you can transfer data only to/from the originating cell.

Tab Context Menu Options

When you click on cells on the **Project Schematic**, some of them open up in separate tabs. The context menu options available depend on which tabs are open. For example, if only the **Project** tab is open, there are no context options available. The following context menu options are available for tabs.

Note

The Project Schematic **Project** tab cannot be closed.

Close Tab

Available for all tabs except for the **Project** tab. Closes the selected tab.

Close Other Tabs

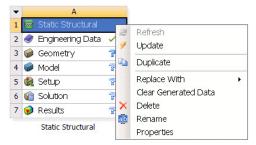
Available for all tabs when multiple are open. If accessed from the **Project** tab, closes all other tabs. If accessed from another tab, closes all tabs except for the one selected and the **Project** tab.

Close All

Available for all tabs except for the **Project** tab. Closes all tabs (including the selected tab) except for the **Project** tab.

System Header Context Menu Options

The following options are available from the context menu for system headers.



Refresh

Refreshes all cells in the selected system.

Update

Updates all cells in the selected system, along with any cells in upstream systems that provide data to them.

Duplicate

Duplicates all cells. No data will be shared between the two systems. Equivalent to issuing a duplicate on the first cell of the system.

Replace with

Enables you to switch the analysis or solver type for an existing system. This functionality is available only from systems that use the Mechanical application. Valid systems that can replace the existing system are listed in the context menu.

Note

If you use **Replace with** to replace a Mechanical system that has a **Section Data** cell with another Mechanical system, the new environment will be added and the original Mechanical environment (including boundary conditions and results) will be deleted.

Delete

Deletes the system from the schematic. You will be prompted to confirm the deletion before any action is taken.

Recreate Deleted Cells

If cells have been removed from the system by a cell-level delete operation or automatically when a self-contained file is loaded into the cell which renders upstream cells unnecessary (such as importing a case file into a CFX setup system), this operation creates new cells at the points where the previous cells were removed.

Rename

Renames the system.

Properties

Displays system properties in the **Properties** view.

Add/Edit Note

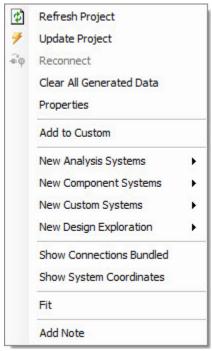
Displays an editable pane where you can enter notes about a system or cell. There is no limit to the amount of text you can type into a note; as you type, the pane increases in length. You can also edit an

existing note by left- or right-clicking on the green triangle in the upper right corner of a system or cell, or by editing the **Notes** field in the **Properties** view. To close the note, click outside the pane. To delete a note, delete the text within the pane or from the **Properties** view.

The content of notes is also included in project reports.

Project Schematic Context Menu Options

Right-click the white space in the **Project Schematic** for the following options. Not all menu options may be visible at all times, depending on the specific configuration of your project.



Resume

Resumes design point updates that were pending when the project was closed. This option is available only if the project has design points that are paused due to cells in a Pending state.

Refresh Project

Refreshes all cells in the project.

Reconnect

Reconnects to updates that were pending when the project was closed. This option is available only if the project has cells in the Pending state.

Upon opening a project, ANSYS Workbench automatically retrieves the data for any completed background update without the need for you to press the **Reconnect** button. If you decline to save the project before exiting, this solver data will be discarded and will not be accessible in future ANSYS Workbench sessions.

Update All Design Points

Performs an update for the selected entry for all design points defined in the **Table of Design Points** (available only if parameters are in use and multiple design points exist).

Update Project

Refreshes input data and generates required output data for all cells in the project. If a project contains multiple design points, this option updates only the current design point.

Note

If you use this option and the project is being updated remotely via RSM, only the systems and cells above the **Parameter Set** bar will be submitted to RSM. If needed, DesignXplorer systems can be further updated once the remote project update is completed. For more information, see Submitting Projects to Remote Solve Manager (RSM) for Remote Update (p. 49).

Add to Custom

You can set up one or more systems with all appropriate connections that represents a project or collection of systems you frequently work with. Use this option to add this collection to your custom templates for easy reuse.

New system type...

Select a new type of system to add to the project. Systems added in this manner will appear as independent systems with no connections to existing systems. The choices are the same as you would see in the **Toolbox**.

Show Connections Bundled

Select this option to show multiple links connecting two systems bundled together in a single link, with a label indicating what cells are connected. For example, if cells 2, 3, and 4 are connected between two systems, the schematic would show a single line connecting the systems, labeled as "2:4" to indicate cells 2-4. This option is off by default. You can toggle between off and on by clicking the option in the context menu.

Show System Coordinates

Select this option to display the system row and column labels. This option is selected by default.

Fit

Select this option to resize the systems in the **Project Schematic** to better fit in the window as it is currently sized.

Add/Edit Note

Displays an editable panel where you can enter notes about a system or cell. There is no limit to the amount of text you can type into a note; as you type, the panel increases in length. You can also edit an existing note by editing the **Notes** field in the **Properties** view. To close the note, click outside the panel. To delete a note, delete the text within the panel or from the **Notes** field in the **Properties** view.

The content of notes is also included in project reports.

Link Context Menu Options

You can right-click a link to see additional context menu options:



Delete

Deletes the selected link.

Properties

Displays detailed information about the link, including the type of link, the origination cell of the link, and the destination cell. Details are shown in the **Properties** view.

Note

To properly update the state of a cell for a linked system, either open the Mechanical application or click the **Update Project** toolbar button.

Release 15.0 - © SAS IP, Inc. All rights reserved Contains proprietary and confidential information	าก
	,,,
of ANSYS. Inc. and its subsidiaries and affiliates.	

ANSYS Workbench Tutor	ıa	IC

To access tutorials for ANSYS Workbench, go to http://www.ansys.com/tutorials.

Release 15.0 - © SAS IP, Inc. All rights reserved Contains proprietary and confidential informatio	'n
of ANŠYS. Inc. and its subsidiaries and affiliates.	

Glossary

analysis system Pre-defined system template that includes all of the cells that are required

to complete a start-to-finish analysis for a particular type of physics, such

as static structural or fluid flow.

archive Save all project files and data into a single package that can be shared,

stored, and reused.

cell Individual part of a system that represents a discrete task in the process

of completing the overall analysis. Typical cells include **Engineering Data**, **Geometry**, **Model** or **Mesh**, **Setup**, **Solution**, and **Results**.

component system Pre-defined system template that includes only the cells necessary to

complete a portion of a complete analysis. Often these systems are task oriented (for example, a system to create a geometry, or produce a mesh),

or are associated with a particular application.

component update An update of a single component (or cell) within a system in an ANSYS

Workbench project. For example, an update of the **Analysis** cell within

a project is a component update.

A **Solution** cell update is the only component update currently

supported by Remote Solve Manager.

context menu A context-sensitive list of options available from a cell or other compon-

ent, accessed via a right-mouse click.

custom system Pre-defined system templates that include all of systems necessary to

complete a coupled analysis, such as FSI or thermal-stress.

data-integrated application An application that has a separate interface from the ANSYS Workbench

project window but that still communicates with ANSYS Workbench.

Design Exploration Systems in the Design Exploration group are used to access DesignXplorer

functionality. These systems connect to the Parameter Set bar in order

to drive parametric studies by varying project parameters.

design point A set of input parameter values and corresponding output parameter

values associated with a single instance of a parameterized project

definition.

design point update

An update of one or more design points in an ANSYS Workbench project.

You can update a single design point, a selected set of design points, or all of the design points in a project. During a design point update, solution data is updated only where output parameters have been defined.

Design point updates can be submitted to Remote Solve Manager.

downstream In the Project Schematic, data flows from top-to-bottom within systems

and from left-to-right between systems. A cell is said to be downstream if it is below a cell in the same system, or to the right of a cell in a separate system. Downstream cells use data from upstream cells as input.

drag-and-drop

An action where you depress the left mouse button over an object, and

while keeping the button depressed, slide the mouse to another location on the interface before releasing the mouse button. Drag-and-drop can be used to introduce systems to the schematic from the Toolbox, to move systems in the schematic, or to form connections between existing

systems in the schematic.

links Systems in the Project Schematic that are dependent on each other in

some manner are connected with links. Links with a square terminator indicate that data is shared between the two cells connected by the link, and links with a round terminator indicate that data is transferred from

the upstream to the downstream cell.

Menu Bar A region of the interface that contains menu options, from which you

can choose various actions, views, and settings.

Parameter An entity that is linked to a data model property within an application.

Parameters can be input, output, or custom.

Project The project is the full collection of systems, components, data, and their

connections that you create to achieve an overall CAE goal.

Project Schematic A region of the ANSYS Workbench project window where you will con-

struct and interact with your project. Projects are represented as connected systems displayed in a flowchart form that allows engineering intent, data relationships, and the state of the analysis project to be understood

at a glance.

project update An update of an entire ANSYS Workbench project. All of the systems,

components, and design points in the project are updated.

refresh An action that reads in all modified upstream data but does not neces-

sarily regenerate the outputs of the cell.

right-mouse click A term meaning to click the right button on the mouse.

systems A collection of cells that together perform a dedicated task. Types of

systems include analysis systems, component systems, and custom sys-

tems.

tab A general term used to indicate an area of the interface where you will

work on your project within the Workbench environment, such as the

Engineering Data or **Parameters** tabs.

Toolbar A region of the ANSYS Workbench project window containing buttons

for quick access to commonly used actions.

Toolbox A region of the ANSYS Workbench interface, located on the left side of

the interface, from which you can choose systems or other components to add to the project. The Toolbox is context-sensitive, meaning the options that appear will change based on what is selected elsewhere in the

interface.

update

An action that updates data in a particular system or cell with any new information that has been added to the project since the last update and regenerates the outputs.

upstream

In the **Project Schematic**, data flows from top-to-bottom within systems and from left-to-right between systems. A cell is said to be upstream if it is above a cell in the same system or to the left of a cell in a separate system. Upstream cells provide their output data to downstream cells.

workflow A visual representation of the objects in your project and their relationship to each other.

Release 15.0 - © SAS IP, Inc. All rights reserved Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.	

Index	chart types, 81
iliuex	chart zoom, pan, and rotate, 85
	saving, 85
,1	using the triad, 85
	viewing, 81
A	clear generated data, 265-266
Add/Edit Note, 266	command window
	journaling, 71
adding a system, 33, 135, 268	component systems, 135, 149, 246
analysis systems, 135, 246	Configuring
building, 38	Workbench, 9
ANSYS Customization Toolkit, 21	console window
ANSYS Workbench	journaling, 71
Configuring, 9	context menu
ANSYS Workbench interface, 245	
tabs, 245	link options, 272
views, 245	Project Schematic options, 271
Appearance options, 15	system header options, 270
Aqwa	tab options, 269
hydrodynamic diffraction analysis, 142	transfer options, 268
hydrodynamic time response analysis, 143	context menus
archiving projects, 71, 75, 259	common options, 265
AUTODYN analysis, 135, 138, 150	custom parameters, 96
	custom systems, 135, 240, 246
В	customizing, 9
BladeGen analysis, 152	_
Blow Molding (Polyflow) analysis, 140	D
component system, 236	data transfer, 33, 43, 268
building a system, 33, 135	defining geometry, 39
	delete, 265
C	deleting a system, 48
CDB files	Design Assessment analysis, 135-136
can be a master mesh in External Data systems, 160	Design Exploration, 242, 246
cells, 245	Design Exploration options, 27
	design points, 93, 101
cell states, 255	activating, 101
common states, 255	activating and exporting, 105
failure states, 257	dpall subdirectory, 73
solution-specific states, 256	exporting, 101
clear generated data, 266	file management, 71
context menu items, 252	running multiple, 101
displaying properties, 266	states, 117
Engineering Data, 252	update order, 104
Geometry, 252	updating, 103
Model/Mesh, 252	using RSM, 107
reset data, 266	viewing a table, 250
Results, 252	DesignModeler to Icepak, 201
Setup, 252	•
Solution, 252	dp0 subdirectory, 72
CFX analysis, 135, 138	dpall subdirectory, 73
component system, 152	duplicate, 265
CFX options, 22	duplicating a system, 46
charts, 49	duplicating systems, 265
chart properties, 82	

E	defining, 39
EKM	Geometry
in ANSYS Workbench, 119	component system, 179
options, 20	Geometry cell, 252-253
Electric analysis, 135, 138	Geometry Import options, 27
Engineering Data	Graphics Interaction options, 16
component system, 153	
Engineering Data cell, 252	Н
errors	Harmonic Response analysis, 135, 141
troubleshooting, 89	help, 87, 266
Excel as a Calculator, 232	Help menu, 265
exiting	Hydrodynamic Diffraction analysis, 135, 142
during RSM design point update, 107	Hydrodynamic Time Response analysis, 135, 143
during RSM Solution cell update, 68	
Explicit Dynamics analysis, 135, 138	1
Explicit Dynamics analysis (AUTODYN), 150	ICEM CFD
Explicit Dynamics analysis (LS-DYNA Export), 153	component system, 179
expressions	Icepak, 197
in parameters, 96	context menu options, 199
Extensions	postprocessing results, 204
options , 21	properties, 201
External Connection Systems, 243	tutorial, 212
External Data component system, 154	imported mesh, 42
External Data systems	importing
CDB files as a master mesh, 160	importing legacy databases, 78
External Model component system, 174	importing files, 71, 74
Extrusion (Polyflow) analysis, 140	importing legacy databases, 259
component system, 236	Internal Combustion Engines, 143
	interrupting an update, 251
F	Iron Python, 71
file management, 71, 74-78	
project directories, 71	J
dp0 subdirectory, 72	journaling, 261
dpall subdirectory, 73	command window, 71
user_files subdirectory, 72	console window, 71
File menu, 259	definition, 69
files	playing a journal, 70
viewing, 247	preferences, 69
Finite Element Modeler	recording, 70
component system, 177	uses, 69
Fluent (with TGrid meshing) analysis	Journals and Logs options, 17
component system, 178	journals and scripts, 69
Fluent analysis, 135, 140	_
component system, 178	L
Fluent options, 23	legacy databases
Fluid Flow analysis, 135, 138, 140	importing, 78
Fluid Flow analysis workflow, 41	licences
FSI analysis, 240	releasing Mechanical during batch runs, 56
	License preferences, 263
G	licenses
generating reports, 18, 262	releasing Mechanical during batch runs, 25
geometry	Licensing, 28

Linear Buckling analysis, 135, 144	Regional and Language options, 16
linking a system, 43	Repository, 20
links	Solution Process, 19
context menu options, 272	
LS-DYNA analysis, 153	P
	parameters, 93, 96
M	chaining, 96-97
Magnetostatic analysis, 135, 144	custom, 96
managing project files, 71	derived parameters, 96
archiving projects, 75	expressions, quantities, and units, 96, 98
file types, 77	input parameters, 96
importing files, 74	output parameters, 96
importing legacy databases, 78	Parameter tab, 96
project locking, 76	viewing, 248
recovering projects, 75	playing a journal, 70
Mechanical analysis workflow, 40	Polyflow analysis, 135, 140
Mechanical APDL	component system, 236
options , 22	postprocessing Icepak results, 204
Mechanical APDL component system, 223	Pre-Stress Modal analysis, 240
Mechanical Model component system, 227	preferences, 13
Mechanical options, 25	journaling, 69
Menu Bar, 245	license, 263
menus, 245	Progress view, 251
Mesh component system, 228	project
Meshing options, 26	archiving, 71, 75, 259
messages	file management, 71
viewing, 251	archiving projects, 75
Microsoft Office Excel component system, 232	importing files, 74
Modal analysis, 135, 145	importing legacy databases, 78
Modal analysis (Samcef), 135, 145	locking projects, 76
Model/Mesh cell, 252-253	recovering projects, 75
moving a system, 48	file types, 77
	importing, 71, 74, 259
N	importing legacy databases, 78
new project, 259	locking projects, 76
	open, 259
0	recovery, 71, 75
open a new project, 259	saving, 259
Options, 13, 263	Project Management options, 14
Appearance, 15	Project Reporting options, 18
CFX, 22	project reports, 86
Design Exploration, 27	Project Schematic, 1, 245
Extensions, 21	context menu options, 271
Fluent, 23	Project tab, 1
Geometry Import, 27	Project Tab, 33
Graphics Interaction, 16	properties, 265-266
Journals and Logs, 17	viewing, 249
Mechanical, 25	Python, 71
Mechanical APDL, 22	•
	Q
Meshing, 26 Project Management, 14	quick help, 265-266
Project Management, 14 Project Reporting, 18	4 a
. reject neporting, re	

R	adding, 33, 135, 268
Random Vibration analysis, 135, 145, 240	component, 149
recently used files, 266	deleting, 48
recording a journal, 70	dependent (connected), 43
recovering projects, 71, 75	duplicating, 46
refresh, 265	duplicating systems, 265
refreshing systems, 266	independent, 43
Regional and Language options, 16	linking, 43
Remote Solve Manager, 49	moving, 48
rename, 265	naming, 37
renaming systems, 266	refreshing systems, 266
replacing a system, 48	renaming, 37, 266
Reports, 18, 262	replacing, 48
Repository	types of, 135
options , 20	updating systems, 265
reset, 265	<u>_</u>
reset data, 266	T
Response Spectrum analysis, 135, 146, 240	tabs, 1, 245
Results cell, 252, 255	Tabs
Results component system, 237	Project Tab, 33
Rigid Dynamics analysis, 135, 146	text files
RSM	importing into Mechanical APDL, 154, 174
submitting Fluids jobs using RSM, 60	thermal results from Icepak to Mechanical, 205
submitting solutions using RSM, 50	Thermal-Electric analysis, 135, 148
submitting Mechanical jobs, 55	Thermal-Stress analysis, 240
	Throughflow analysis, 148
S	Toolbar
saving, 259	File menu, 259
scripting, 261	Help menu, 265
definition, 71	Tools menu, 263
setting journaling preferences, 69	Options, 13
setting license preferences, 263	Units menu, 264
Setup cell, 252, 254	View menu, 262
single license sharing, 28	Views
Solution cell, 252, 254	messages, 251
Solution Process	outline view, 248
options , 19	progress, 251
solve manager, 50	properties, 249
states, 255	table, 250
common states, 255	viewing files, 247
design points, 117	Toolbox, 1, 9, 245-246
failure states, 257	Tools menu, 263
solution-specific states, 256	transfer data, 268
Static Structural analysis, 135, 147	Transient Structural, 148
Static Structural analysis (Samcef), 135, 147	Transient Structural analysis , 135
Steady-State Thermal analysis, 135, 147	Transient Thermal analysis, 135, 149
surface data	troubleshooting, 89
format for External Data, 161	TurboGrid analysis, 238 Tutorial
system categories, 245	
System Coupling	Icepak in Workbench, 212 Tutorials, 275
component system, 237	types of parameters, 96
systems, 135, 149, 240, 246	types of parameters, 30

U units base and common units, 11 configuring, 10 custom unit systems, 13 in parameters, 96 menu, 264 predefined unit systems, 12 selecting, 264 update, 265 updating design points, 103 activating and exporting, 105 update order, 104 using RSM, 107 updating systems, 265 user_files subdirectory, 72 using ANSYS Workbench, 49, 246 adding a system, 33 building analyses, 38 charts, 49 deleting a system, 48 duplicating a system, 46 journals and scripts, 69 linking a system, 43 moving a system, 48 naming systems, 37 parameters and design points, 93 project reports, 86 Remote Solve Manager, 49 replacing a system, 48 using RSM, 50 submitting Mechanical jobs, 55 using RSM to submit Fluids jobs, 60 Using ANSYS Workbench adding a system, 135 using the command window journaling, 71 view all/customize, 246 View menu, 262 views, 1, 245, 262 parameters and design points, 93 viewing a table of design poitns, 250 viewing charts, 81 viewing files, 247 viewing messages, 251 viewing progress, 251 viewing properties, 249 viewing the outline, 248 Vista AFD analysis, 238

Vista CCD analysis, 238

Vista CPD analysis, 239 Vista RTD analysis, 239 Vista TFanalysis, 239

W

window layout, 262 workflow Fluid Flow analysis, 41 Mechanical analysis, 40 workspaces, 1